Winter 11-25-2012

Proceeding of International Conference on Advances in Mechanical and Industrial Engineering

DR. CH R. VIKRAM KUMAR

Follow this and additional works at: https://www.interscience.in/conf_proc_volumes

Part of the Other Mechanical Engineering Commons
Advances in Mechanical and Industrial Engineering deals with recent developments and practices adopted in various projects in different engineering disciplines and specializations - Rock Dredging; Concrete Technology; Grid Computing; Electric Propulsion & the Stationary Plasma Thruster; Turbo charging; Ultra filtration, Nan filtration & Reverse Osmosis; FACTS Devices; Sensors; Advanced Materials for Aircraft and Helicopters; Data Communication and Network Protocol; Satellite Communication Systems; Optoelectronic Devices; Wireless Communication; Applications of CFD Techniques in Aero-propulsive Characterization of Missiles; Hazardous Waste Management; Liquid Fueled SCRAMJET Combustors; Armor Materials and Designs; Heat Transfer in NuclearReactors; Defense Electronics Systems; World Class Manufacturing; Value Engineering & Engineering Ethics. Mechatronics is an interdisciplinary branch of mechanical engineering, electrical engineering and software engineering that is concerned with integrating electrical and mechanical engineering to create hybrid systems. In this way, machines can be automated through the use of electric motors, servo-mechanisms, and other electrical systems in conjunction with special software. A common example of a Mechatronics system is a CD-ROM drive. Mechanical systems open and close the drive, spin the CD and move the laser, while an optical system reads the data on the CD and converts it to bits. Integrated software controls the process and communicates the contents of the CD to the computer.

Robotics is the application of Mechatronics to create robots, which are often used in industry to perform tasks that are dangerous, unpleasant, or repetitive. These robots may be of any shape and size, but all are preprogrammed and interact physically with the world. To create a robot, an engineer typically employs kinematics (to determine the robot's range of motion) and mechanics (to determine the stresses within the robot).

Robots are used extensively in industrial engineering. They allow businesses to save money on labor, perform tasks that are either too dangerous or too precise for humans to perform them economically, and to ensure better quality. Many companies employ assembly lines of robots, especially in Automotive Industries and some factories are so robotized that they can run by themselves. Outside the factory, robots have been employed in bomb disposal, space exploration, and many other fields. Robots are also sold for various residential applications, from recreation to domestic applications.

The field of mechanical engineering can be thought of as a collection of many mechanical engineering science disciplines. Several of these sub disciplines which are typically taught at the undergraduate level are listed below, with a brief explanation and the most common application of each. Some of these sub disciplines are unique to mechanical engineering, while others are a combination of mechanical engineering and one or more other disciplines. Researchers are working on applying their wireless and mobile research to transportation, health care, education, collaboration and environmental sustainability. Projects already underway include safe and efficient road transportation, autonomous driving, wireless medical implants, mobile video delivery, multiparty wireless videoconferencing and energy harvesting.

The Conference sometimes is conducted in collaboration with other institutions. IRNet encourage and invite proposals from institutes within India to join hands to promote research in various areas of discipline. These conferences have not only promoted the international exchange and cooperation, but have also won favorable comments from national and international participants, thus enabled IRNet to reach out to a global network within three years time. The conference is first of its kind and gets granted with lot of blessings.

The conference designed to stimulate the young minds including Research Scholars, Academicians, and Practitioners to contribute their ideas, thoughts and nobility in these disciplines of Engineering. IRNet received a great response from all parts of country and abroad for the presentation and publication in the proceeding of the conference.

I sincerely thank all the authors for their invaluable contribution to this conference. I am indebted towards the reviewers and Board of Editors for their generous gifts of time, energy and effort. It’s my pleasure to
welcome all the participants, delegates and organizer to this international conference on behalf of IRNet family members. We in IRNet believe to make “RESEARCH COOL”.

I wish all success to the paper presenters. The papers qualifying the review process will be published in the forthcoming IOAJ journal.

Convenor :-
Rahul K Prasad
“FINITE ELEMENT ANALYSIS OF PARTITIONED VERTICAL STORAGE COLUMN”

SUJIT P. PATIL¹, M. V. KAVADE ², U. M. NIMBALKAR³, VINAAY PATIL⁴

¹, ², ³ Mechanical Engineering Dept., Rajarambapu Institute of Technology, Sakharale, Islampur, Maharashtra, India; 
⁴ FEA (Finite Element Analysis) Dept, Vaftsy CAE, Pune, Maharashtra, India

Abstract- A vertical storage column is typically used to store liquids or fine powders. To maximize the storage capacity, these columns are usually very tall. They are typically undergoes to wind loads, as the bending caused is much greater at these heights. A partitioned vertical column is the one in which separate compartments are made so that different types of material can be stored in the single column leading to better space utilization. In this project work three compartments are made in the cylindrical pressure vessel by adding three partitioned plates at an angle of 120° with each other and then FEA analysis is done by considering one of the compartments is fully occupied with design pressure conditions. This paper investigates the deformation and equivalent von-mises stresses of partitioned vertical column with nonlinear static structural analysis.

Keywords- Finite Element Analysis [FEA], Pressure vessel, Vertical storage column

I. INTRODUCTION
A pressure vessel is a closed container designed to hold gases or liquids at a pressure substantially different from the ambient pressure. The pressure differential is dangerous and many fatal accidents have occurred in the history of their development and operation. In a partitioned vertical column separate compartments are made for storing different types of material in the single column. Typically the restriction on such columns is that, all the materials to be stored need to be at the same pressure, this ensure structural stability of the system. The design problem is to achieve the same structural stability, even when the materials are stored at different pressures. This work will focused to study on Optimization and analysis of Partitioned Vertical Storage Column. As we will able to store different materials in one Partitioned Vertical Storage Column instead of storing these materials in different vertical storage column it results in huge cost saving. This work will definitely gives reliable data for the design of Partitioned Vertical Storage Column.

II. PROCESS INFORMATION
Nitrous Oxide is obtained by ammonium nitrate pyrolysis synthesis. The following reaction indicates thermal decomposition of ammonium nitrate to gives nitrous oxide.

• \[\text{NH}_4\text{NO}_3 \rightarrow 2 \text{H}_2\text{O} + \text{N}_2\text{O} \]

It is an exothermic reaction occurring at around 200 deg Celsius. Nitrous Oxide thus generated has a lot of impurities Ammonium Nitrate fumes, Nitrogen, Other oxides of Nitrogen, Steam. The impurities are removed by washing or Scrubbing, which is a three stage process 1st Stage: Steam is condensed in this stage by water Scrubbing 2nd Stage: Residual traces of Ammonium Nitrate are removed by caustic Scrubbing 3rd Stage: Caustic Scrubbing generates Ammonia gas which is removed by a acid Scrubbing.

III. PROPOSED SOLUTION
The objective behind the Partitioned Column is to have stages 5, 6, 7, and 8 shown in the fig. 01 in a single vessel.

IV. LITERATURE SURVEY-
As there are lot of work is done in design of pressure vessel, very little work is done in relevant field of partitioned vertical column. A brief review of some selected references on evaluation and strength of partitioned vertical column.

Fig01. Block diagram of process
Fig02 Partitioned vertical storage column
Suraj Alage, Prof S. V. Diwan [2] has suggested that the cylindrical pressure vessel be split into 2 compartments and a partition plate is used for segregation of the vessel. However the partition plate becomes a critical component in design, if it gets subjected to a differential pressure. Comparative evaluation of straight & curved split in partitioned pressure vessel has to be done to know performance of vessel under different pressure conditions. 

Rama Subba Reddy Gorla [3] In the present work, an LNG storage tank has been computationally simulated and probabilistically evaluated in view of the several uncertainties in the fluid, structural, material and thermal variables that govern the LNG storage tank. A finite element code ALGOR was used to couple the thermal profiles with structural design. The stresses and their variations were evaluated at critical points on the storage tank. Cumulative distribution functions and sensitivity factors were computed for stress responses due to fluid, mechanical and thermal random variables. These results can be used to quickly identify the most critical design variables in order to optimize the design and make it cost effective.

T. Aseer Brabin, T. Christopher, B. Nageswara Rao [4] have presented study on Finite element analysis (FEA) has been carried out to obtain the elastic stress distribution at cylinder-to cylinder junction in pressurized shell structures that have applications in the design of aerospace pressure vessels.

V. SCOPE OF PROJECT-
This project aims at design and analysis of the proposed model of the vertical storage column to find out stress and deflection in its various components using FEA and then optimizing the weight and shape of the vertical storage column using iterative FEA. The complete project runs through the following steps:

i. Study of pressures of various working conditions coming on Partitioned Vertical Storage Column

ii. FEA of existing Vertical Storage Column. From the nonlinear static structural analysis of existing Vertical Storage Column the area of maximum and minimum stress have been located.

iii. Optimize the result of the partitioned storage column in the form of thickness and area of partitioned compartments in vessel.

iv. Comparison of the basic design and the optimized model for conditions of dimensions of parts, weight.

Procedure for Analysis-
To determine the stresses when one of the compartments is fully occupied to design pressure conditions. [1]

Steps involved in analysis of Partitioned Vertical Storage Column-

- Creating CAD geometry in Pro-E
- Importing CAD geometry in ANSYS 12.0
- Meshing of partitioned pressure vessel
- Applying boundary conditions
- Solving for result in terms of stress & deformation

VI. SPECIFICATION-

<table>
<thead>
<tr>
<th>Name of Component</th>
<th>Dimensions (MM)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter of Vessel External</td>
<td>9.00E+03</td>
</tr>
<tr>
<td>cylinder (d1)</td>
<td></td>
</tr>
<tr>
<td>Diameter of Vessel Internal</td>
<td>4.00E+03</td>
</tr>
<tr>
<td>cylinder (d2)</td>
<td></td>
</tr>
<tr>
<td>Vessel Vertical Height</td>
<td>2.10E+04</td>
</tr>
<tr>
<td>Vessel Skirt Height</td>
<td>3.00E+03</td>
</tr>
<tr>
<td>Vessel Top</td>
<td>132</td>
</tr>
<tr>
<td>Vessel Base</td>
<td>132</td>
</tr>
<tr>
<td>Vessel Skirt Support Thickness</td>
<td>102</td>
</tr>
<tr>
<td>Partition Wall Thickness</td>
<td>102</td>
</tr>
<tr>
<td>Partition Wall Angle</td>
<td>120° Each</td>
</tr>
<tr>
<td>Design Temperature</td>
<td>300K</td>
</tr>
<tr>
<td>Vessel External cylinder thickness</td>
<td>102</td>
</tr>
<tr>
<td>Vessel Internal cylinder thickness</td>
<td>102</td>
</tr>
<tr>
<td>Yield Strength</td>
<td>240Mpa</td>
</tr>
<tr>
<td>Factor of safety</td>
<td>1.5</td>
</tr>
<tr>
<td>Ultimate Strength</td>
<td>320Mpa</td>
</tr>
</tbody>
</table>

Table 01 Dimensions and Properties for Partitioned Vertical Column

VII. MODELLING & MESHING-
Fig.03 shows the 3-D model of partitioned vertical column. It is created in modeling software Pro-E then exported to ANSYS Workbench, which is required for the purpose of further analysis. ANSYS Workbench provides a highly integrated engineering simulation platform, supports multi-physics engineering solutions and provides bi-directional parametric association with most available CAD systems.

The filter sheet assembly model is meshed with 20 Node Hex-Dominant SOLID 186 element. It is a higher order 3-D 20-node solid element that exhibits quadratic displacement behaviour. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z
directions. The element supports plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elasto-plastic materials, fully incompressible elastic materials.

VIII. ANALYSIS-

While doing FEA of partitioned vertical column, design pressure conditions applied to one of the compartment, also the whole assembly is fixed at bottom of skert support as shown in fig.05

The maximum stress (von-mises) in the Vessel Top cover (80mm) and Vessel Base cover (80mm) are below Yield Strength. Further proceeding to analysis of External cylinder and internal cylinder.
The max stress for the external cylinder (60mm) and internal cylinder (60mm) thickness are not exceeding the safe limit value i.e.160Mpa. Further taking FEA for partitioned wall, considering the thickness value for all three partitioned wall as 80mm the maximum Equivalent stress (Von-Mises) is near 160Mpa. That means they are not exceeding the safe limit value i.e.160Mpa.

Fig.10 Von-Mises Stress Plot

Fig.11 Total Deformation Plot

Fig. no 10 &11 shows stress & deformation occurred in Partitioned Vertical Storage [8]

**IX. RESULTS-**

<table>
<thead>
<tr>
<th>SR. NO</th>
<th>DIMENSIONS OF COMPONENTS</th>
<th>RESULT (VON-MISES STRESS)</th>
<th>WEIGHT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>102mm internal vessel cylinder thickness</td>
<td></td>
<td>66.42</td>
</tr>
<tr>
<td>2</td>
<td>102mm external vessel cylinder thickness</td>
<td></td>
<td>51.57</td>
</tr>
<tr>
<td>3</td>
<td>102mm partitioned flat plate no-01</td>
<td></td>
<td>75.25</td>
</tr>
<tr>
<td>4</td>
<td>102mm partitioned flat plate no-02</td>
<td></td>
<td>67.17</td>
</tr>
<tr>
<td>5</td>
<td>132mm Vessel Top</td>
<td></td>
<td>34.61</td>
</tr>
<tr>
<td>6</td>
<td>132mm Vessel Base</td>
<td></td>
<td>39.27</td>
</tr>
<tr>
<td>7</td>
<td>60mm internal vessel cylinder thickness</td>
<td></td>
<td>153.09</td>
</tr>
<tr>
<td>8</td>
<td>60mm external vessel cylinder thickness</td>
<td></td>
<td>109.27</td>
</tr>
<tr>
<td>9</td>
<td>80mm partitioned flat plate no-01</td>
<td></td>
<td>127.44</td>
</tr>
<tr>
<td>10</td>
<td>80mm partitioned flat plate no-02</td>
<td></td>
<td>112.62</td>
</tr>
<tr>
<td>11</td>
<td>80mm Vessel Top</td>
<td></td>
<td>75.19</td>
</tr>
<tr>
<td>12</td>
<td>80mm Vessel Base</td>
<td></td>
<td>77.5</td>
</tr>
</tbody>
</table>

Table 02 Result table for Analysis of Partitioned Vertical Storage

**X. CONCLUSION-**

Structural optimization for components of partitioned vertical storage column is achieved from initial weight of the partitioned vertical storage 9.086e+005 kg to final weight of 5.549e+005 kg. The weight optimized for the partitioned pressure vessel is 3.537e+005 kg. It gives overall reduction in weight by 38.92%. The basic objective of this project is to design and analysis of proposed model of partitioned pressure vessel assembly using FEA. However the partition plate becomes a critical component in design, if it is gets subjected to a differential pressure. Also the interference between the vessel and partition is a critical area and need to analyze in FEA to understand effects on stress attributes of the vessel.

**Future Scope-**

As future work, the design of pressure vessels when two compartments are fully occupied to design pressure conditions and when all three compartments are fully occupied to design pressure conditions. These issues are presently under consideration.

**REFERENCES-**

[1] Proprietary information of POWER GAS INC, PGI USA.
[2] Suraj Alage, Prof S. V. Diwan, “FEA based Comparative Evaluation of Straight Split Vs Curved Split in Partitioned Pressure Vessel”.
[7] ANSYS 12.0 software version
Abstract:-Earth-to-air heat exchangers, also called ground tube heat exchangers, are an interesting technique to reduce energy consumption in a building. Several papers have been published in which a design method is described. In this paper, long tube is introduced for Earth-tube effect, area required and improving performances. Outside fresh air is drawn from a filtered air intake. Earth-tube smooth-walled rigid or semi rigid plastic attached with tube, below 1.5 to 3 m (5 to 10 ft) underground where the ambient earth temperature is typically 10 to 23 °C (50-73 °F) all year round in the temperate latitudes where most humans live. Fan pulls air from a tank through a short tube. Earth tubes are often a viable and economical alternative or supplement to conventional air conditioning systems since there are no compressors, chemicals or burners and only blowers are required to move the air. Earth-air heat exchangers can be very cost effective in both up-front/capital costs as well as long-term operation and maintenance costs. In the context of today's diminishing fossil fuel reserves, increasing electrical costs, air pollution and global warming, properly designed earth cooling tank-tube offer a sustainable alternative to reduce or eliminate the need for conventional compressor-based air conditioning systems, in non-tropical climates.

Keywords: Earth-to-air heat exchanger; passive cooling; soil types; and tube.

1. INTRODUCTION

As the need for energy-efficient building designs increases, the use of passive heating/cooling and renewable resources also increases. One way to reduce the use of energy in the heating and cooling of ventilation air is to preheat the air in the winter and pre-cool the air in the summer using an earth-air heat exchanger (EAHE) also known as an earth-tube. In an EAHE, ventilation air is drawn into the building through a system of tubes located in the soil near or beneath the building. EAHEs are not a new technology; indeed, the concept dates back at least to the 1st century BC in the Middle East (Oleson, 2008), where air was cooled in the Qantas that were used to transport water and also used in Roman architecture. The EAHE concept is quite simple: a tube is buried in the soil as shown in Figure 1. The soil will be at a temperature warmer than the outside air in winter and cooler than the outside air in summer.

Ventilation air is drawn into the building through the buried tube, heating it in the winter and cooling it in the summer—and passively reducing the overall cooling and heating load. In the past two decades, much research has been conducted to develop analytic and numerical methods for analyzing EAHEs (Hollmuller 2003; Florides and Kalogirou 2007; Tittelein et al. 2009; Lee and Strand 2008; Ghosal et al. 2005; Cucumo et al. 2008).

2. CONCEPT DESCRIPTIONS

The principle of earth to air underground heat exchangers is very simple. The system uses the seasonal thermal storage ability of the soil, which has a temperature delay compared to the outdoor temperature. This temperature difference between the outdoor temperature and the soil temperature enables a cooling effect of the hot summer air and a heating effect of the cold winter air. Figure 5.2-1 shows that the deeper the heat exchanger is situated, the larger is the active temperature difference, which can be attained between the outdoor temperature and the earth temperature. Therefore, the heat exchanger should be placed as deep as possible. The costs for the excavation depend on the level, on which the heat exchanger is placed, and these costs influence the choice of depth.

The utilization of the stored heat or cold in the ground with the means of an underground heat exchanger depends on the ground composition and the local microclimate. The solar global radiation
heats the surface of the earth and the heat absorbed depends on the surface character (cover, construction, etc.). The temperature of the ground in the different layers depends on the climate (outdoor temperature, wind, precipitation, etc.) and the composition of the earth (heat conductivity, specific heat and density). Since the underground heat exchanger operates close to the earth surface, there is no influence from geothermal parameters. The underground heat exchanger should be placed on ground water level. The ground temperature is influenced if the heat exchanger is placed in areas with dense construction area or under a building. The heat exchanger can only be applied in climates with big temperature differences between summer and winter and between day and night.

Typical arrangements for the pipes are:

i. Parallel piping directly under the foundation or between the single and continuous strip foundation
ii. Laying the piping in the foundation ditch around the building
iii. Laying the piping in ditches in the surrounding yard

The distance between the pipes should be approximately 1.0 meter. If the pipes are placed closer than 1 meter to each other, the interaction between the pipes will be too strong. More than 1 meter distance does not bring extra benefit for the discharging cycles. If the system is installed under a building, cold basement facilities must be considered.

The required size of the heat exchanger depends on the designed air volume rate and on the available space. Applying a small system to achieve comfort increase in housings could be relatively inexpensive. The inlet and outlet of the heat exchanger can be established with simple standard components. Larger systems and systems placed in ground water are much more complex since the air inlet and outlet can only be achieved through a plenum duct. A maximum air velocity of 2 m/s is recommended for smaller systems (larger systems can be designed for air velocities up to 5 m/s). This corresponds approximately to 250 m³/h for the commonly applied synthetic pipes with a diameter of 20 cm. More exact values can be seen in Figure 5.2-4. The optimal piping length is found through the pipe diameter and the air velocity, which depends on the pressure drop in the system. Large diameters only make sense at very long piping.

In order to properly design a cooling pump (GHP) system, it is important to know the seasonal variation in soil temperature. These soil thermal properties depend strongly on soil porosity and moisture content. Therefore, any preliminary assessment of a potential GHP project will require knowing the soil texture and the average groundwater level at the project site. The amplitude of seasonal changes in soil temperature on either side of the mean earth temperature depends on the type of soil and depth below the ground surface. At depths greater than about 30 feet below the surface, however, the soil temperature remains relatively constant throughout the year, as shown in Figure 3, below.

Vertical closed-loop earth heat exchangers are installed in boreholes 200 to 300 feet deep, where seasonal changes in soil temperature are completely damped out. Well-based open-loop systems also extend to this depth or deeper. These ground loop configurations are thus exposed to a constant year-round temperature. On the other hand, horizontal-loop, spiral-loop, and horizontal direct-expansion (DX) loops are installed in trenches that usually are less than 10 feet deep. For these types of ground loops, it is important to accurately know the expected seasonal changes in the surrounding soil temperature. The extra cost of installing such systems in deeper trenches may be outweighed by the gain in thermal performance, since deeper soils have less pronounced seasonal temperature changes and are thus closer to room temperature, which reduces the work load of the heat pump units.
Deeper soils not only experience less extreme seasonal variations in temperature, but the changes that do occur lag farther behind those of shallower soils. This shifts the soil temperature profile later in the year, such that it more closely matches the demand for heating and cooling. Referring to Figure 4 for example, the maximum soil temperature occurs in late August (when cooling demand is high) at a depth of 5 feet below the ground surface, but occurs in late October (after the heating season has begun) at a depth of 12 feet below the surface.

Thus a deeper ground loop installation would lower the annual operating cost for electrical energy to run the heat pumps, and over the life of a GHP system, these accumulated savings may more than offset the higher capital cost of burying the ground loop more deeply. In order to determine the optimal depth of burial, it is important to accurately know how the seasonal change in soil temperature varies with depth, which is mainly determined by the soil's thermal properties.

### 3. SOIL THERMAL PROPERTIES

Heat capacity (also known as specific heat) indicates the ability of a substance to store heat energy; the greater its heat capacity, the more heat it can gain (or lose) per unit rise (or fall) in temperature. The heat capacity of dry soil is about 0.20 BTU per pound per °F of temperature change, which is only one-fifth the heat capacity of water. Therefore, moist or saturated soils have greater heat capacities, typically in the range of 0.23 to 0.25 BTU/lb/°F. As shown in Figure 3 above, light dry soils experience greater seasonal temperature swings at a given depth than wet soils. This is because their lower heat capacity causes their temperature to rise or fall more than wet soils for a given amount of heat energy gained in the spring or lost in the fall.

### 4. A WORKABLE SOLUTION

In order to specify a workable solution designer of ETHE systems should evaluate:

- **Location**: If the system is predominantly cooling you will want the collector in a permanently shaded area near a lake or river. If the system is predominately heating it should be located in a sunny area without some aquifer competing to steal away the heat.

- **Depth of pipe (temperature)**: Available shortwave radiation on the collector surface is directly related to the mode the system favors, most with consideration for the depth of pipe and can typically be between 1.5m to 3m. A system that is designed predominately for cooling in an area without shading will need the pipes buried deeper than a system designed for heating in the same locale.
c. Soil conductivity (heat transfer): Dry sand is the worst thing one would want in the process of conducting heat to and from buried ducts. Dense, wet and conductive (can you say moist compacted clay) is the trick.
d. Duct material and tube connections: Options include concrete, metal, plastics – with or without conductive fins or antimicrobial agents. The ducts cannot sag under loading. The connections have to be robust and of the highest quality as they and the ducts have to deal with ground moisture and soil gases. They must be of the most conductive material for the lowest cost with the least air flow resistance but offer the best characteristics against corrosion.
e. Temperatures, flow, velocity, diameter, friction, length, layout and drainage (the thermal to hydraulic part of the calculation): According to application engineers, velocities between 6m/s to 10m/s are typical. This means the diameter has to be picked based on flows and friction losses. To optimize designs based on the thermal and pressure requirements, using several shorter lengths in parallel in a reverse return arrangement can in some cases be better than a single longer serpentine loop or it may be more suitable to use a loop that follows the perimeter of the buildings foundation.
f. Air entering the ground duct and the HVAC system will need to be conditioned: This means it has to be suitable for inhalation by the occupants through the decontamination of particulates, moisture, odours, gases and biological concerns.
g. Energy analysis: The designer will need to evaluate the capital and operating costs of the system, including the electricity to run the fan and decontamination equipment to assure the energy used is lower than the cooling or heating power offered by the system.
h. Building science issues: Depending on the type of system there could be issues with infiltration and short circuiting of the ground exchanger. It is necessary that both building and ducts are sealed tightly to prevent differential pressures across and within the home from interfering with the required differential pressures in the ETHE.

5. CONCLUSION

The paper presented results of a study of the potential of the earth-air heat exchanger for reducing indoor temperature in buildings. It has demonstrated the impact of the various parameters on the thermal behaviour of the earth-air heat exchanger. The EAHX has been evaluated within TRNSYS environment to study the performance of the earth-air system and its impact on the indoor environment.

Application
In cooling modus, the heat exchanger is suitable for independent cooling of indoor air as well as for the supply of another cooling system. Possibilities for cooling are natural night ventilation, mechanical night ventilation and building mass activation. Three applications of cooling with an underground heat exchanger are “comfort cooling”, “room cooling” and “supplement cooling”.

Benefits
Lower cooling energy costs, hygienically controlled air input (lower concentration of bacteria and fungi spores in the inlet air), possibility to reduce or avoid a conventional cooling system.

Typical cost indicators (relative to a conventional HVAC system)

- Operating costs – lower
- Operating maintenance costs – lower
- Investment costs - higher

The underground heat exchanger can be combined with a conventional AC system, with a considerable reduction of its cooling load. Further, this technology can be combined with other passive cooling technologies such as night ventilation or can be used as a pre-stage to a heat pump.

REFERENCES

[2]. Fachgebiet Bauphysik und Solarenergie, Universität Siegen, Germany: http://nesal.uni-siegen.de
[3]. Huber Energietechnik AG, Zürich, Switzerland: http://www.igjzh.com/huber/wkm/wkm.htm
DESIGN OF ROTATE LEFT&RIGHT LOGIC BASED BIDIRECTIONAL BARREL SHIFTER

P.MURALI KRISHNA & CH.RAJENDRAPRASAD
SR engineering college

Abstract:-reverse ble operation is important thing in quantum computing, which is emerging technique. This paper will present the design of the barrel shifter that performs logical shift right, arithmetic shift right, rotate right, logical shift left, arithmetic shift left, and rotate left operations. The main objective of the upcoming designs is to increase the performance without proportional increase in power consumption. In this regard reversible logic has become most popular technology in the field of low power computing, optical computing, quantum computing and other computing technologies. Rotating and data shifting are required in many operations such as logical and arithmetic operations, indexing and address decoding etc. Hence barrel shifters which can shift and rotate multiple bits in a single cycle have become a common choice of design for high speed applications. The design has been done using reversible fredkin and feynman gates. In the design the 2:1 mux can be implemented by fredkin gate which reduce quantum cost, number of ancilla bits and number of garbage outputs. The feynman gate will remove the fanout. By comparing the quantum cost, number of ancilla bits and number of garbage outputs the design is evaluated. We can extend this technique for addition also.

Keywords- barrel shifters, quantum cost, ancilla bits, verilog.

I. INTRODUCTION

Rotating and shifting data is required in several applications including variable-length coding, arithmetic operations, and bit-indexing. Consequently, barrel shifters, which are capable of shifting or rotating data in a single cycle, are commonly found in both digital signal processors and general purpose processors. In reversible system information is not erased. Thus in reversible gates number of inputs and outputs are equal which means that the input stage can always be retained from the output stage. If a bit is erased in an irreversible circuit then it will dissipate kTln2 joules of heat energy where k is the Boltzmann’s constant and T is the absolute temperature of environment [4]. There won’t be dissipation of kTln2 joules of heat energy if the operations are performed in reversible manner based on reversible logic circuits [3]. Based on this observation, Bennett [3] showed, for a reversible computer the heat dissipation is exactly kTln1 which is logically zero. Thus reversible computation is a highly potential field for upcoming low power/high performance computing. Reversible logic also has the applications in emerging nanotechnologies such as quantum dot cellular automata, quantum computing, optical computing and low power computing, etc.

The constraints involved in designing reversible circuits using reversible gates are:

a. The fan-out of every signal is equal to one.
b. Loops are not permitted in a strictly reversible system.

On the other hand, data shifting and rotating is important and frequently used in arithmetic operations, bit-indexing, variable-length coding and many more. The reversible circuits have associated overhead in terms of number of garbage outputs and the number of ancilla inputs. The outputs which do not perform any useful operation and needed to maintain reversibility of the circuit are termed as garbage outputs, while an auxiliary constant input used to design a reversible circuit is called the ancilla input bit [17].

A (n,k) barrel shifter is a combinational circuit with n inputs and n outputs where k select lines controls the shift operation. The existing designs of the reversible barrel shifters can only perform the left rotate operation [11], [15].

The reversible barrel shifter can shift and rotate multiple bits in a single cycle and thus will be considerably faster than the reversible sequential shift register. This paper examines design alternatives for barrel shifters that perform the following operations: shift right logical, shift right arithmetic, rotate right, shift left logical, shift left arithmetic, and rotate left.

In the proposed design the basic building blocks are reversible Fredkin and Feynman gates.

The structure of the paper is as follows: Section II provides the necessary background on reversible logic as well as the definitions of some commonly used reversible logic gates. Section III describes several barrel shifters. Section IV describes the proposed design of the reversible bidirectional barrel shifter. In Section V, the performance analysis of the proposed shifter is presented. Lastly, the conclusions and further studies are discussed in Section VI

II. REVERSIBLE GATES

A Reversible Gate is an n-input, n-output (denoted by n * n) circuit. To maintain the reversibility property of reversible logic gates several dummy output signals are needed to be produced in order to equal the number of input to that of output. These
signals are commonly known as **Garbage Outputs**. For example, for reversible Exclusive-OR operation Feynman gates are used which produce an extra dummy output along with its principal output signal to preserve reversibility. The quantum cost of reversible gate is equal to the number of 1x1 and 2x2 reversible gates needed to design a 3x3 reversible gate. The quantum cost of all 1x1 and 2x2 reversible gates are considered as unity [18], [7], [2]. The 3x3 reversible gates are designed from 1x1 NOT gate, and 2x2 reversible gates such as Controlled-V and Controlled-V+ (V is a square-root of NOT gate and V+ is its hermitian), the Feynman gate which is also known as Controlled NOT gate.

A NOT gate is 1x1 gate represented as shown in Fig. 1. Its quantum cost is unity since it is a 1x1 gate.

![Fig. 1. NOT GATE](image)

The input vector, $Iv$ and output vector, $Ov$ for 2x2 **Feynman Gate (FE)** is defined as follows: $Iv = (A, B)$ and $Ov = (P = A \bar{B} + A \bar{C})$. Feynman gates are typically used as copying gates. If $Iv = (A, B=0)$ then $Ov = (P =A \bar{B})$. Fanout is not allowed in reversible logic. Feynman gate is helpful in this regard as it can be used for copying the signal by which it avoids the fanout problem as shown in Fig.2(c).

![Fig. 2. CNOT Gate, its quantum implementation and its useful properties](image)

The input and output vector for 3x3 **Fredkin gate (FR)** [1] are defined as follows: $Iv = (A, B, C)$ and $Ov = (P =A \bar{B} + A \bar{C} \land R = A \bar{C} \land AB)$. Figure 3(a) shows the block diagram of a Fredkin gate. A Fredkin gate can work as 2:1 MUX, as it is able to swap its other two inputs depending on the value of its first input. The first input A works as a controlling input while the inputs B and C work as controlled inputs as shown in the Fig. 3(a). Thus when $A=0$ the outputs P and Q will be directly connected to inputs A and B and if $A=1$ the inputs B and C will be swapped resulting in the value of the outputs as $Q=C \land R=B$. The quantum implementation of a Fredkin gate with a quantum cost of 5 is shown in Figure 3(b) [7]. In Fig. 3(b) each dotted rectangle is equivalent to a 2x2 Feynman gate and the quantum cost of each dotted rectangle is considered as 1 [18]. The same assumption is used for calculating the quantum cost of the Fredkin gate [7]. Thus, the quantum cost of the Fredkin gate is 5 as it consists of 2 dotted rectangle, 1 Controlled-V gate and 2 CNOT gate.

![Fig. 3. Fredkin Gate and its quantum implementation](image)

### III. BIDIRECTIONAL BARREL SHIFTER

A barrel shifter is a combinational circuit which has n-input and n-output and m select lines that controls bit shift operation. A barrel shifter having n inputs and k select lines is called (n,k) barrel shifter. Barrel shifter can be unidirectional allowing data to be shifted or rotated only to left (or right), or bidirectional which provides data to be rotated or shifted in both the directions. The logarithmic barrel shifter is most widely used among the different designs of barrel shifter, because of its simple design, less area and the elimination of the decoder circuitry. An n-bit logarithmic barrel shifter has a total of log2(n) stages. Each stage determines whether to shift or not to shift the input data. The stage k will shift the input $2^k$ times if the control bit $sk$ (where $k=0, 1, ... (log2(n)-1)$ ) is set to 1 otherwise the input will remain unchanged. Logarithmic shifter is more efficient in terms of design as well as area but delay cost is large [11]. This paper presents the designs of reversible bidirectional arithmetic and logical barrel shifter that can perform six operations: logical right shift, arithmetic right shift, right rotate, logical left shift, arithmetic left shift and left rotate. The existing shifter is a unidirectional logarithmic shifter consists of multiplexers. A 3x3 Fredkin Gate works as simple (2:1) multiplexers. Feynman gates are used for producing fanouts.

The existing shifter is complex in design and requires large number of gates. As a result the total number of garbage outputs is high. Thus there is great room for improving the circuit complexity, total number of gates and garbage outputs, delay and quantum cost. For efficient designing of a reversible circuit several criteria are needed to be considered:

a. Minimize the number of gates as possible.
b. Minimize the quantum cost of the circuit.
c. Total number of garbage outputs and usage of constant inputs should be minimized.
By maintaining the above parameters and observing the previous design, a novel logarithmic Reversible Barrel Shifter has been proposed. The proposed barrel shifter is a left rotating shifter which uses Fredkin gates for producing fan outs. A (4, 2) logarithmic barrel shifter has been illustrated in Figure 4. The circuit uses a total of 6 Fredkin gates, 4 Feynman gates and produces 6 Garbage outputs. The Quantum cost of the circuit has also been evaluated. The calculation shows that the Quantum Cost of the proposed (4, 2) circuit is 34.

![Figure 4. Proposed Reversible (4, 2) Barrel Shifter](image)

### IV. DESIGN OF REVERSIBLE BIDIRECTIONAL BARREL SHIFTER

The proposed design of reversible bidirectional barrel shifter can perform logical right shifting, arithmetic right shifting, rotating right, logical left shifting, arithmetic left shifting and rotating left operations. The proposed reversible bidirectional arithmetic and logical barrel shifter design approach is illustrated as shown in Fig. 5 with an example of a (8,3) barrel shifter. The barrel shifter performs the various operations such as logical right shift, logical left shift, rotate left etc. depending on the values of sra, sla, rot and left control signals. Table I shows that for different values of control signals sra, sla, rot and left the operations that can be performed by a (8,3) reversible bidirectional arithmetic and logical shifter.

<table>
<thead>
<tr>
<th>Operation performed</th>
<th>Control signal values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Logical right shift</td>
<td>Left=0 Rot=0 Sra=0 Sla=0</td>
</tr>
<tr>
<td>Arithmetic right shift</td>
<td>Left=0 Rot=0 Sra=1 Sla=0</td>
</tr>
<tr>
<td>Rotate right</td>
<td>Left=0 Rot=1 Sra=0 Sla=0</td>
</tr>
<tr>
<td>Logical left shift</td>
<td>Left=1 Rot=0 Sra=0 Sla=0</td>
</tr>
<tr>
<td>Arithmetic left shift</td>
<td>Left=1 Rot=0 Sra=1 Sla=0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Operation</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>3-bit shift right logical</td>
<td>0 0 0 a7a6a5a4a3</td>
</tr>
<tr>
<td>3-bit shift right arithmetic</td>
<td>a7a7a7a6a5a4a3</td>
</tr>
<tr>
<td>3-bit rotate right</td>
<td>a2a1a0a7a6a5a4a3</td>
</tr>
<tr>
<td>3-bit shift left logical</td>
<td>a4a3a2a1a0 0 0 0</td>
</tr>
<tr>
<td>3-bit shift left arithmetic</td>
<td>a7a3a2a1a0 0 0 0</td>
</tr>
<tr>
<td>3-bit rotate left</td>
<td>a4a3a2a1a0a7a6a5</td>
</tr>
</tbody>
</table>

The design of a reversible barrel shifter can be divided into six modules: (i) Data reversal control unit-I, (ii) Arithmetic right shift control unit, (iii) Shifter or rotation unit which consists of three sub-modules that performs Stage I, Stage II and Stage III operations, (iv) Rotation unit, (v) Arithmetic left shift control unit, (vi) Data reversal control unit-II. The reversible design of the modules of the reversible bidirectional barrel shifter along with their working are explained as follows:

1. **Data Reversal Control Unit-I**

In reversible barrel shifter the direction of the shift operation performed is controlled by the control signal left as shown in the Table I. The reversible bidirectional barrel shifter performs the shift operation in the left direction if the value of control signal left as 1, that is, the arithmetic left shift operation or logical left shift operation. Otherwise, the shift operation is performed in the right direction for the value of left=0, that is, arithmetic right shift operation or logical right shift operation. The data reversal control unit-I has Fredkin gates, since two outputs of the Fredkin gate can work as 2:1 MUXes. 4 Fredkin gates can be used to reverse the 8 bit input data by utilizing two outputs of the Fredkin gate as 2:1 Muxes. A left shift operation for a n bit input data by k-bit can be performed in three steps:

(i) reverse the input data,

(ii) perform k bit right shift operation, and

(iii) reverse the outputs of the step (ii).

For example, for a 8-bit input data i7, i6, i5, i4, i3, i2, i1, i0 the three steps of logical left shift operation...
by 3 bits will be: (i) reverse i7, i6, i5, i4, i3, i2, i1, i0 to produce i0, i1, i2, i3, i4, i5, i6, i7, (ii) perform the 3 bit logical right shift operation to produce 0, 0, 0, i0, i1, i2, i3, i4, and (iii) reverse the outputs of step (ii) to yield i4, i3, i2, i1, i0, 0, 0, 0. The date reversal control unit-I is shown in Fig. 5.

2 Arithmetic Right Shift Control Unit

The reversible arithmetic right shift control unit is shown in Fig. 5. The arithmetic right shift operation is controlled by the arithmetic right shift control unit. The designing of this unit is done using a single Fredkin gate controlled by the control signal sra, and preserves the sign bit of input data. The arithmetic right shift operation is performed if the value of control signal sra = 1, otherwise it simply passes the data to the next module. Multiple copies of the sign bit are created using the Feynman gates because fanout is not allowed in reversible logic.

![Fig. 5. Proposed (8,3) reversible bidirectional barrel shifter](image)

*FE represents Feynman Gates, FR represents Fredkin gates and G represents the garbage outputs

3 Shifter Or Rotation Unit

The three stage design of the reversible shifter or rotation unit is as shown in Fig. 5. The amount of shift operation that has to be performed is done by the shifter unit in the design of reversible bidirectional barrel shifter. This unit is controlled by the control signals S2, S1 and S0. This unit can be divided into three stages. Depending on the value of control signal S2, S1 and S0, the first, second and the third stages of this unit right shifts the input data by 2², 2¹ and 2º bits respectively. All the three stages are designed using the chain of 8 Fredkin gates controlled by the control signals S2, S1 and S0. The Feynman gates are used in the design to avoid the fanout problem. The working of the three stages of the shifter unit is explained as follows:

- **Stage – I**: The first stage of shifter unit is controlled by the control signal S2 and it will shift the input data by 2²-bits. The input data is right shifted by 2²-bits if the value of control signal S2 is 1, else the input data remains unchanged. The outputs of the Stage I is passed as inputs to Stage II of the shifter unit.
- **Stage – II**: The second stage of the shifter unit is controlled by the control signal S1 and it works on the outputs of the first stage. The input data provided to the second stage is right shifted by 2¹-bits if the value of control signal S1 is 1, else the input data remains unchanged. The outputs of the Stage II is passed as inputs to Stage III of the shifter unit.
- **Stage – III**: The third stage of the shifter unit is controlled by the control signal S0. The output data generated by the stage-II is right shifted by 2º-bits if the value of control signal S0 is 1 else the output data remains unchanged. The outputs of this stage is passed as inputs to the next module in the design of reversible bidirectional barrel shifter.

4 Rotation Unit

The rotation unit is shown in Fig. 5. The rotation operation is controlled by the rotation unit. The designing of this unit is done using a chain of 8 Fredkin gates and controlled by the control signal rot, and performs the rotation operation of input data. The rotation operation is performed if the value of control signal rot = 1, otherwise it simply passes the data to the next module.

5 Arithmetic Left Shift Control Unit

The arithmetic left shift control unit is shown in Fig. 5. The design of the arithmetic left shift control unit and the design of the arithmetic right shift control unit are same. This control unit is controlled by the control signal sla and is responsible to perform the arithmetic left shift operation. This unit is implemented using a single Fredkin gate. This unit preserves the sign bit needed to perform the arithmetic left shift operation if the value of control signal sla = 1, else it simply passes the LSB of the shifter or rotation unit.

6 Data Reversal Control Unit II

The data reversal control unit is controlled by the control signal left. If the value of control signal left is 1, this unit reverses its input data to generate a left shifted result else it simply passes the input data to its outputs. The data reversal control unit II reverses its 8 bit input which consists of 1 bit from the output of the arithmetic left shift control unit and 7 bits from the outputs of the shifter unit. The design of this unit is
shown in Fig. 5 which is same as explained for data reversal control unit I.

The (8,3) reversible bidirectional arithmetic and logical barrel shifter uses 32 Feynman gate to copy the input data to avoid the fanout, and 41 Fredkin gates are used for arithmetic and logical bidirectional shifting and rotating. The above design of the (8,3) reversible bidirectional barrel shifter can be generalized to design a (n,k) reversible bidirectional barrel shifter.

V. PERFORMANCE ANALYSIS

To avoid the fanout problem, in the proposed design Feynman gate is used. Chains of n/2 Fredkin gates are used in data reversal unit-I and data reversal unit-II. The arithmetic right shift control unit uses one Fredkin and $2^k-1$ Feynman gates. Chain of n Fredkin gates and n Feynman gates are used in shifter or rotation unit at each stage. Rotation unit $2^k$ fredkin gates for m=0 to (k-1) for each stage. One Fredkin gate and one Feynman gate is used in arithmetic left shift control unit. Thus the total number of Fredkin gates used to design a (n,k) reversible bidirectional barrel shifter can be written as:

$$FR= \text{Number of Fredkin gates used in data}$$
$$\text{reversal control unit-I}+ \text{Number of Fredkin gates used}$$
$$\text{in arithmetic right shift control unit}+ \text{Number of Fredkin gates used in rotation}$$
$$\text{unit-I}+ \text{Number of Fredkin gates used in arithmatic}$$
$$\text{left shift control unit}+ \text{Number of Fredkin gates used in data}$$
$$\text{reversal control unit-II}=n/2+1+(n*k)+1+n/2$$

$$=\sum_{m=0}^{k-1} 2^m+n*(k+1)+2.$$

The total number of Feynman gates used to design a (n,k) reversible bidirectional barrel shifter is:

$$FE= \text{Number of Feynman gates required to design}$$
$$\text{arithmetic right shift control unit}+ \text{number of}$$
$$\text{Feynman gates used in shifter or rotation unit}+$$
$$\text{number of Feynman gates used in arithmetic}$$
$$\text{left shift control unit}=2^n-1+(n*k)+1.$$

A. Ancilla input Bits

The table III shows the number of ancilla bits required to design a reversible bidirectional barrel shifter for different values of n and k. $2^k+(n*k)$ Feynman gates are required to design a (n,k) reversible bidirectional barrel shifter. Each Feynman gate requires one ancilla input bit to copy the input data. Additionally, the Fredkin gate used in arithmetic shift control unit requires one ancilla bit. Hence the total number of ancilla inputs (ANs) required to design a (n,k) reversible bidirectional arithmetic and logical barrel shifter is $ANs=2^k+(n*k)+1$. Table III shows that the total number of ancilla inputs required to design a (8,3) reversible bidirectional barrel shifter are 33 which is same as illustrated in Fig. 5.

**TABLE III**

<table>
<thead>
<tr>
<th>n/k</th>
<th>n=4</th>
<th>n=8</th>
<th>n=16</th>
<th>n=32</th>
<th>n=64</th>
</tr>
</thead>
<tbody>
<tr>
<td>K=2</td>
<td>13</td>
<td>21</td>
<td>37</td>
<td>69</td>
<td>133</td>
</tr>
<tr>
<td>K=3</td>
<td>33</td>
<td>57</td>
<td>105</td>
<td>201</td>
<td></td>
</tr>
<tr>
<td>K=4</td>
<td>81</td>
<td>145</td>
<td>273</td>
<td></td>
<td></td>
</tr>
<tr>
<td>K=5</td>
<td>193</td>
<td>353</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>K=6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>449</td>
</tr>
</tbody>
</table>

B. Quantum Cost

Table IV shows the quantum cost for a reversible bidirectional barrel shifter for different n and k values. The number of Feynman and Fredkin gates used will decide the quantum cost of (n,k) reversible bidirectional barrel shifter. The quantum cost of the Feynman gate is considered as one, while the quantum cost of the Fredkin gate is considered as five. Hence the quantum cost of the proposed design of (n,k) reversible bidirectional barrel shifter can be calculated as Quantum Cost = 5 * (number of Fredkin gates) *(number of Feynman gates).

The quantum cost(QC) of the (n,k) reversible bidirectional barrel shifter can be represented as

$$QC=5 \times (\sum_{m=0}^{k-1} 2^m+n*(k+1)+2)+ 2^k+(n*k).$$

The quantum cost of a (8,3) reversible bidirectional barrel shifter shown in Fig. 5 is 237.

**TABLE IV**

<table>
<thead>
<tr>
<th>n/k</th>
<th>n=4</th>
<th>n=8</th>
<th>n=16</th>
<th>n=32</th>
<th>n=64</th>
</tr>
</thead>
<tbody>
<tr>
<td>K=2</td>
<td>137</td>
<td>165</td>
<td>301</td>
<td>573</td>
<td>1117</td>
</tr>
<tr>
<td>K=3</td>
<td>237</td>
<td>421</td>
<td>789</td>
<td>1525</td>
<td></td>
</tr>
<tr>
<td>K=4</td>
<td>565</td>
<td>1029</td>
<td>1957</td>
<td></td>
<td></td>
</tr>
<tr>
<td>K=5</td>
<td></td>
<td>1317</td>
<td>2437</td>
<td></td>
<td></td>
</tr>
<tr>
<td>K=6</td>
<td></td>
<td></td>
<td>3013</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

C. Garbage Outputs

For different reversible bidirectional barrel shifter designs the number of garbage outputs produced is as
shown in Table V. In the table, n is the number of input data bits and k represents the shift value. In the design of (n,k) reversible bidirectional barrel shifter the shifter unit can be designed in k stages and each stage consists of the chain of n Fredkin gates to perform the shift operation. Each Fredkin gate in the chain of n Fredkin gates produces at least one garbage output except the last Fredkin gate which produces two garbage outputs. Two garbage outputs are produced by Fredkin gate which is used in the design of arithmetic left shift control unit and arithmetic right shift control unit. One garbage output is produced by last Fredkin gate of the data reversal control unit-II as the control signal left cannot be utilized further. Hence the number of garbage outputs (GOs) required to design a (n,k) reversible bidirectional arithmetic and logical shifter can be written as \( \text{GOs} = k(n + 1) + 6 + \frac{n^2 - 2n + 2}{4} \). In (8,3) reversible bidirectional barrel shifter design the number of garbage outputs produced in Fig. 5 are 40 which is equal to the result in Table V.

**TABLE V**

<table>
<thead>
<tr>
<th>n/k</th>
<th>n=4</th>
<th>n=8</th>
<th>n=16</th>
<th>n=32</th>
<th>n=64</th>
</tr>
</thead>
<tbody>
<tr>
<td>K=2</td>
<td>19</td>
<td>27</td>
<td>43</td>
<td>75</td>
<td>139</td>
</tr>
<tr>
<td>K=3</td>
<td>40</td>
<td>64</td>
<td>112</td>
<td>208</td>
<td></td>
</tr>
<tr>
<td>K=4</td>
<td>89</td>
<td>153</td>
<td>281</td>
<td></td>
<td></td>
</tr>
<tr>
<td>K=5</td>
<td></td>
<td></td>
<td>202</td>
<td>362</td>
<td></td>
</tr>
<tr>
<td>K=6</td>
<td></td>
<td></td>
<td></td>
<td>459</td>
<td></td>
</tr>
</tbody>
</table>

**VI. CONCLUSIONS**

In this paper An Efficient Design of Reversible Logic Based Bidirectional Barrel Shifter has been proposed. The design of the proposed bidirectional shifter is done using Fredkin gates and Feynman gates. The number of garbage outputs, the number of ancilla inputs and the quantum cost of the (n,k) reversible bidirectional barrel shifter increase more rapidly by varying n and keeping k as a constant compared to the designs in which n is kept as a constant while k is varied. The functional verification of the proposed design of the reversible barrel shifters are performed through simulations using the Verilog HDL flow for reversible circuits. The design of bidirectional barrel shifter is been evaluated in terms of garbage outputs, ancilla inputs and the quantum cost. The proposed design of reversible bidirectional barrel shifter can perform logical right shifting, arithmetic right shifting, rotating right, logical left shifting, arithmetic left shifting and rotating left operations.

REFERENCES

HALF-EFFECT SOLAR POWERED ABSORPTION COOLING SYSTEM USING AMMONIA-SALT

AJOY DEBBARMA

M.Tech Scholar, Mechanical Engineering (Thermal), National Institute of Technology, Mechanical Engineering, Assam, Silchar, 788010, India

Abstract:- Thermal-driven refrigeration systems have attracted increasing research and development interest in recent years. These systems do not cause ozone depletion and can reduce demand on electricity. The main objective of this work is to perform theoretical concept of a thermal-driven refrigeration system using a new sorbent-sorptive pair as the working pair. The active component of sorbent is sodium thiocyanate (NaSCN), lithium nitrate (LiNO3). Ammonia (NH3) is chosen as sorptive. Among the working pairs compared, NH3/LiNO3 showed the best results. By using a two stage half-effect LiNO3 or NaSCN absorption chiller instead of single effect chiller, it is possible to decrease the required generator temperature of the chiller and this allowed the use of simple flat plate collectors or evacuated tube collectors to produce hot water to be used as the hot source for absorption chiller.

Key Words: Half-Effect system, Absorbent-Refrigerant pairs, solar flat plate collectors.

1. INTRODUCTION

Refrigeration and air conditioning systems have a major impact on energy demand with roughly 30% of total energy consumption in the world [1]. The advantage of a solar cooling system is related to primary energy savings from the use of free solar energy. Most of the absorption chillers require a high driving temperature, as high as 80–110ºC. Some water-cooled absorption chillers can be driven with a lower temperature heat but generally, the COP of the chiller then drops sharply. Since solar radiation is not always high enough to drive such high-temperature driven systems with low-cost collectors, many systems are now equipped with highly efficient but expensive collectors. This capital investment, consequently, is quite often so high as to counteract primary energy savings from solar-driven operation [2].

Two types of the absorption chillers, the single effect and half effect cycles, can operate using low temperature hot water. Absorption chillers that can provide relatively high coefficients of performance (COP) require high temperature heat sources. Typical COPs for single-effect absorption chillers are 0·7 to 0·8, while double-effect chillers produce COPs of 1·0 to 1·2. However, to achieve this level of performance, a heat supply temperature of 80ºC to 110ºC will be required for single-effect chillers, while double-effect chillers will require a temperature of around 150ºC. By using a one or two stage half-effect absorption chiller instead of single effect chiller, it is possible to decrease the required generator temperature of the chiller and this allowed the use of simple flat plate collectors to produce hot water to be used as the hot source for absorption chiller. Therefore solar collectors are more efficient with half-effect system than that with single effect system. The principle of the half-effect cycle is that it has one or two lifts. The term lift is used to represent a concentration difference between the generator and absorber. This concentration difference is what drives or gives the potential for mass to flow into the absorber. With the single effect there is only one lift.

The binary systems of NH3-H2O and LiBr-H2O were well known as working fluid pairs. The advantage for refrigerant NH3 is that it can evaporate at lower temperatures (i.e. from -10 to 0°C) compared to H2O (i.e. from 4 to 10°C). Therefore, for refrigeration, the NH3-H2O cycle is used. Research has been performed for NH3-H2O systems theoretically and experimentally. These studies show that the NH3-H2O system exhibits a relatively low COP. It is proposed that NH3-LiNO3 and NH3-NaSCN cycles can be alternatives to NH3-H2O systems [4]. Both of the pairs don't require rectification. And in particular, NH3/LiNO3 was proven to start generation of refrigerant vapour at much lower temperature (Infante Ferreira, 1984; 1985) [5].

2. DESCRIPTION OF THE HALF-EFFECT SYSTEM

It must be noted that, any absorption refrigeration system can be operated only when the solution in the absorber is richer in refrigerant than that in the generator. When the temperature increases or the pressure reduces, the fraction of refrigerant contained in the solution is reduced, and vice versa. When the generator temperature is dropped, the solution circulation rate will be increased causing the COP to drop. If it is too low, the system can be no longer operated. The half-effect absorption system was introduced for an application with a relatively low-temperature heat source. [6]

Absorption chillers that can provide relatively high coefficients of performance (COP) require high...
temperature heat sources. Typical COPs for single-effect absorption chillers are 0.7 to 0.8, while double-effect chillers produce COPs of 1.0 to 1.2. However, to achieve this level of performance, a heat supply temperature of 80°C to 100°C will be required for single-effect chillers, while double-effect chillers will require a temperature of around 150°C. A half-effect absorption cycle is a combination of two single-effect cycles but working at different pressure levels. Letting heat source temperature be lower than the minimum temperature is necessary for a single-effect cycle working at the same pressure level.

The major difference between the single and half effect cycle is the addition of a second absorber with or without second generator. This addition allows for an extra operating pressure at the low Concentration [LC] absorber and low temperature [LT] generator. The principle of the half effect cycle is that it has one or two lifts. The term lift is used to represent a concentration difference between the generator and absorber. This concentration difference is what drives or gives the potential for mass to flow into the absorber. With the single effect there is only one lift and thus as the hot water temperature is decreased the difference between the two concentrations decreases. The one or two lifts enable the half-effect cycle to operate at lower firing generator temperatures. It can provide cooling with a low-temperature heat source. The cooling COP is roughly half of the SE (Single-Effect) cycle as it rejects more heat than a single-effect absorption cycle around 50% and is called so HE (Half-Effect) cycle.

\[
\text{COP} = \frac{\text{Cooling capacity obtained at evaporator}}{\text{Heat input for the generator + work input for the pump}}
\]

2.2 Double-Lift Half-effect Absorption System.
Double-lift half-effect absorption cycle is a combination of two single-effect cycles but working at different pressure levels. This cycle has two solution circuits. Refrigerant from the evaporator goes through two solution pumps to the condenser. This is why it is called a two-stage or double-lift cycle.

The two stage half-effect absorption refrigeration system as shown in Fig.2 consists of condenser, evaporator, two generators, two absorbers, two pumps, two solution heat exchangers, two solution reducing valves and a refrigerant expansion valve. [7].

Two stages half effect cycle takes heat input at two pressure levels. The condenser and evaporator operate at high and low pressures. The vapour generated at an intermediate pressure is fed to a second absorber, which feeds the generator at high pressure. Both generators can be supplied with heat at the same temperature. Fig. 2(a) represents the schematic arrangement of the half effect vapour absorption refrigeration system. The system consists of two solution circuits, each consisting of an

---

The refrigerant vapour from the evaporator is absorbed by the weak solution in the high absorber. The strong solution from the high absorber is pumped to the high generator through the high solution heat exchanger. The weak solution in the high generator is returned to the high absorber through the high solution heat exchanger. The refrigerant is boiled out of the solution in the high generator and circulated to the condenser. The liquid refrigerant from the condenser is returned to the evaporator through an expansion valve. Both generators are supplied with heat at the same temperature. Fig. 2(b) shows the half effect vapour absorption refrigeration cycle on a log p-T diagram. The cycle 1–2–3–4–5–6 represents solution circuit 1, and the cycle 8–9–10–11–12–13 represents solution circuit 2. [1] Fig. 2(c) shows the half effect vapour absorption refrigeration cycle on a log p-T diagram cycle without RHEx.

The circulation ratio of the low pressure stage can be obtained using the refrigerant mass balance for the low absorber. The circulation ratio was obtained from the equation,

\[ CR_L = \frac{1-x_4}{x_1-x_4} \]  

Similarly, the circulation ratio for the high pressure stage is given by

\[ CR_H = \frac{1-x_{11}}{x_9-x_{11}} \]  

The heat input to the cycle is

\[ Q_{in} = Q_{EV} + Q_{LPG} + Q_{HPG} - Q_{loss} \]

The heat rejected in the cycle is

\[ Q_{out} = Q_{CO} + Q_{LA} + Q_{HA} - Q_{gain} \]
The actual coefficient of performance is calculated with out considering the heat loss from the generators.

\[ \text{COP}_a = \frac{Q_{eV}}{(Q_{LPG} + Q_{HPG})} \]  

The real coefficient of performance is calculated taking into account the heat loss from the generators.

\[ \text{COP}_r = \frac{Q_{eV}}{(Q_{LPG} + Q_{HPG} - Q_{loss})} \]  

The second law efficiency is

\[ \eta_{II} = \frac{\text{COP}_a}{\text{COP}_c} \]  

The ideal absorption cycle can be considered as a combination of a Carnot driving cycle and reversed Carnot cooling cycle. The uncertainties in the calculation of COPa, COPr and second law were ±3.2% and ±4.6% and ±6.2%, respectively. [11]

3. REFRIGERANT-ABSORBENT WORKING PAIRS

Performance of an absorption refrigeration system is critically dependent on the chemical and thermodynamic properties of the working fluid. The mixture should also be chemically stable, non-toxic, and non-explosive [6].

The binary systems of NH3-H2O and LiBr-H2O were well known as working fluid pairs. The advantage for refrigerant NH3 is that it can evaporate at lower temperatures (i.e. from -10 to 0°C) compared to H2O (i.e. from 4 to 10°C). Among different options of working fluids that can be used as alternative to NH3-H2O are NH3-LiNO3, NH3-NaSCN. Both of the pairs doesn’t require rectification. And in particular, NH3/LiNO3 was proven to start generation of refrigerant vapour at much lower temperature (Infante Ferreira, 1984; 1985).

Figure 3 shows the comparison of COP values vs. generator temperatures for NH3-H2O, NH3-LiNO3 and NH3-NaSCN absorption cycles. There exists a low generator temperature limit for each cycle. Each cycle cannot be operated at generator temperatures lower than its limit. For the NH3-LiNO3 cycle a lower generator temperature can be used than for the others. It is shown that, for low generator temperatures, the NH3-LiNO3 cycle gives the best performance.

Further comparison, performances of these three cycles against various, evaporator, condenser and absorber temperatures are compared. The results show that with the increase in evaporator temperature, the COP values for each cycle increase. For evaporator temperatures lower than zero, which is the temperature range for refrigeration, the NH3-NaSCN cycle gives the best performance.

For condenser temperatures ranging from 20°C to 40°C, both the NH3-NaSCN and NH3-LiNO3 cycles show better performance than the NH3-H2O cycle.

The effect of absorber temperature is similar to that of condenser temperature. The results show that the system performance for the NH3-NaSCN and NH3-LiNO3 cycles is better than that for the NH3-H2O cycle, not only because of higher COP values, but also because of no requirement for analysers and rectifiers. Therefore, they are suitable alternatives to the ammonia-water cycle. [4].

4. SOLAR COLLECTORS

Solar collectors are more efficient with half-effect system than that with single effect system; this can be explained by the fact that single effect system need higher generator temperature than that required by half-effect system.

Three different collectors were considered. The first one is a vacuum tube collector and the other two were flat plate collectors.

Table 1

<table>
<thead>
<tr>
<th>Type</th>
<th>Efficiency*</th>
<th>Dimension</th>
<th>LxWxH</th>
</tr>
</thead>
<tbody>
<tr>
<td>Collector I</td>
<td>Vacuum tube</td>
<td>0.523/0.87/0.0023</td>
<td>2.39x0.73x0.12</td>
</tr>
<tr>
<td>Collector II</td>
<td>Flat</td>
<td>0.723/2.65/0.0100</td>
<td>1.27x2.45x0.11</td>
</tr>
<tr>
<td>Collector III</td>
<td>Flat</td>
<td>0.682/4.30/0.0077</td>
<td>1.94x1.03x0.08</td>
</tr>
</tbody>
</table>

\[ \eta_{sol} = \eta_{0} - \alpha_{1} T_{m} - \alpha_{2} T_{m}^{2} \]  

*\( \eta_{sol} \) = \( \eta_{0} - \alpha_{1} T_{m} - \alpha_{2} T_{m}^{2} \), \( T_{m} = (tm-ta)/I \), Gross efficiency with wind.

Specifications are summarised in Table 1. Their efficiency curves can also be found in Fig. 5. Collectors were chosen from an information source (SPF, www.spf.ch). Collector I and II were selected because they were best in performance. Collector III
was moderate in performance and presumed to serve as a more economic model.[5]

Comparing cooling efficiency with the three different collector’s vacuum collector I, Collector II, Collector III, it can be seen from Fig.7, the Half Effect system show only higher efficiencies in Collector II.

5. CONCLUSIONS
In this paper, an attempt has been made to study the combination: flat plate solar collectors and half effect absorption cooling systems with Ammonia-salt. The main results obtained are concluded below:

- By using a half-effect LiNO₃ absorption chiller instead of single effect chiller, it is possible to decrease the required generator temperature of the chiller and this allowed the use of simple flat plate collectors to produce hot water to be used as the hot source for absorption chiller.
- The half-effect absorption unit can be found in applications where the heat source temperature is too low to be used to fire a single-effect unit. The advantage of the half-effect cycle is that the heat-input temperature can be less than the single-effect for the same evaporator and heat rejection temperature.
- The low-temperature-driven absorption cycles discussed here use the NH₃/LiNO₃ working pair, which is the general choice for low-temperature heat recovery applications.
- Solar collectors are more efficient with half-effect system than that with single effect system.
- The half-effect cycle can achieve a higher capacity compared to the single-effect cycle over a larger range of low temperature and low flow rate applications.

The COP of the half-effect cycle, 0.35, is roughly half of the single-effect COP, 0.7. Therefore, the half-effect cycle is only practical when a large amount of low-grade waste heat that cannot be utilized by a single-effect cycle is available.

The advantage of the half-effect cycle over the single-effect cycle occurs if the heat source is free and it is lower than 200°F or has a low flow rate.

REFERENCES


[2]. D.S. Kim*, C.H.M. Machielsen**. Comparative study on water and air cooled solar absorption cooling systems. * Phone: +31 (0)15 278 69 78, fax: +31 (0)15 278 24 60, d.s.kim@wbmt.tudelft.nl  ** Phone: +31 (0)15 278 44 55, fax: +31 (0)15 278 24 60, c.h.m.machielsen@wbmt.tudelft.nl


[7]. M.H. Madveshi, P.N. Gupta, Nitin Pal. Use of Solar Energy for Absorption Cooling System to Drive Half-Effect.


[15]. Z. CREPINSEK, D. GORICANEC, J. KROPE. Comparison of the performances of absorption refrigeration cycles. WSEAS TRANSACTIONS on HEAT and MASS TRANSFER, 2000
Abstract: A CFD based analysis of laminar natural convection flow and heat transfer in inclined air cavity with top wall heating was carried out. The variation of Nusselt number with inclination of cavity of different aspect ratios was investigated. The present results confirm that the Nusselt number for the cavity at any inclination can be obtained by a simple scaling of the Nusselt number for the vertical cavity. It holds good for a wide range of aspect ratios.

Keywords- inclined cavity, top wall heating, natural convection, CFD.

I. INTRODUCTION

Natural convection flow and heat transfer in enclosed regions has been of considerable interest since long time. One such example is natural convection in a rectangular enclosure whose vertical walls are at different temperatures (vertical cavity). This temperature difference causes fluid motion inside the enclosure and consequently heat is transferred from the hot wall to the cold wall. The enclosure can act as a means of insulation for buildings, double glazed glass windows, glass doors, industrial furnaces and chimneys, and many other heat transfer equipments. In these applications, natural convection in the enclosure limits its insulating effect. Apart from numerous applications of vertical enclosure, there are practical situations when the enclosure is inclined to the gravity vector (inclined cavity). It is of practical importance in skylights, roof windows, solar energy flat plate collectors and other solar energy applications. The inclined cavity offers two very different flow situations; one in which the hot wall is at the bottom (bottom wall heating case - BWH) and second in which the hot wall is on the top (top wall heating case - TWH). Interestingly, there has been extensive research on bottom wall heating case as it presents very interesting aspects of flow instability (for a horizontal cavity, flow instability occurs when the Rayleigh number exceeds a critical value of about 1708). There have been many excellent review papers on natural convection in vertical and inclined cavities with bottom wall heating in the past. In recent past, Fomichev et al. [1] and Henderson et al. [2] have reviewed some of these investigations particularly relevant to applications in building construction like fenestration products and solar water heater. Hollands and Konicek [3], Ozoe et al. [4] and Elsherbiny et al. [5] have investigated heat transfer by natural convection across vertical and inclined air layer at different inclinations. Holland et al. [6] investigated experimentally natural convection heat transfer rates for inclined air cavity of high aspect ratios heated from below.

On the other hand, for a horizontal cavity with top wall heating, the top wall is maintained at higher temperature than the bottom wall and so the density no longer decreases in the direction of gravitational force. Conditions are now stable and there is no bulk fluid motion in the horizontal cavity. The heat transfer occurs from top to bottom and it is due to conduction. However, as soon as the cavity is inclined to the horizontal direction, the flow circulation starts and natural convection heat transfer takes place from hot wall to cold wall. There are many practical applications in which top wall heating condition appears in inclined cavity especially in building fenestration. Arnold et al.[7] investigated the boundary layer regimes for inclined rectangular cavities and showed that for sufficiently high Ra, a rescaling of the results for vertical cavity could be applied for inclined cavity in certain range of inclinations. The lack of investigations for moderate values of Rayleigh number in this area has prompted us to carry out this CFD based study of laminar natural convection in inclined air cavity with top wall heating (TWH case).

II. PROBLEM STATEMENT

Fig. 1 shows an inclined cavity with top wall heating. In the figure, H is the height of the cavity (dimension of hot/cold wall) and L is the width of the cavity (lateral spacing between the hot and cold wall). The inclination of the cavity is defined by the angle θ, the hot wall makes with the hot/cold wall and Tc. The end walls separating the hot wall and cold wall are assumed to be adiabatic. Apart from inclination θ, there are three non-dimensional parameters associated with the flow, namely, Rayleigh number (Ra = $\frac{g\beta HT^3}{\nu\alpha}$), Prandtl number (Pr = $\frac{\nu}{\alpha}$) and aspect ratio ($A = \frac{H}{L}$). In the present work, the value of Ra was kept constant and computations were carried out for different A and θ. All computations were carried out using a commercial CFD code “Fluent”. The range of parameters covered is as follows: Rayleigh number Ra = 1.87×10^5 (Fixed), fluid – Air Aspect ratio A = 7, 14, 21, 28.57, 35 Inclination θ = 0°, 15°, 30°, 45°, 60°, 75°, 90°
III. COMPUTATIONAL DETAILS

The fixed Ra value was obtained by keeping the temperature difference AT between the hot and cold walls as 5 K (T_h=308 K and T_c=303 K) and the air properties constant at mean temperature of 305.5 K. The value of Ra selected is a representative value for laminar natural convection in applications like fenestration products and solar water heater. The other two end walls were kept as adiabatic. The value of width of the cavity L was kept constant at 0.035m in all cases and different aspect ratios A were obtained by appropriately changing the height H of the cavity. For the 2-D cavity geometry, the uniform grid in both directions was preferred as it facilitates resolution of flow equally well throughout the flow domain (i.e. near the walls as well as in the core region). The physical size of the grid cell was kept same in all aspect ratios; thereby the number of grids depended upon the aspect ratio of the cavity. The grid sizes were 49×16 for A=7, 98×16 for A=14, 147×16 for A=21, 200×16 for A=28.57 and 245×16 for A=35.

The steady state laminar natural convection flow in 2-D cavity was solved by setting the options in Fluent as Boussinesq approximation for buoyancy force, pressure based solver for solution procedure, SIMPLE algorithm for pressure-velocity coupling and second order upwind schemes for discretization. In the present computations, every individual case was initialized by the similar initial field and the convergence criterion was set to values of residues as initialized by the present computations, every individual case was obtained.

IV. RESULTS AND DISCUSSION

The computed flow field results are discussed here with a typical streamline plot and a corresponding isotherm plot. Fig. 2 shows the streamline plot in cavity of aspect ratio A=14 at θ = 30°. At 0° i.e. horizontal cavity with TWH, the stagnant fluid is in stable equilibrium and there is no fluid motion. This is true for all aspect ratios. But, as soon as the cavity is tilted the fluid motion starts. There is global unicellular flow (Primary circulation) in the cavity. The fluid near hot wall flows up the hot wall and the fluid near the cold wall flows down the cold wall, thus resulting in a unicellular primary circulation in the cavity. This flow feature continues unchanged at all inclinations. The intensity of circulation, of course, increases with increase in θ. The flow is similar in other aspect ratio cavities. However, in some cases, small secondary cells appear embedded in the centre core region of the primary circulation. This is seen in high aspect ratio cavities at inclinations 0°-60°. The weak secondary cells continue to exist right into the vertical cavity 0°-90°.

The constant temperature lines are perfectly horizontal and parallel in the horizontal cavity (θ=0°). It is a stable temperature gradient field and causes no fluid motion. With increase in θ, isotherms start crowding near bottom half of the hot wall and top half of the cold wall. The crowding of isotherms indicates increase of temperature gradient on the walls and represents the formation of thermal boundary layer type flow on the walls as shown in Fig. 3 for cavity with A=14 and θ=30°.

The heat transfer results are presented in terms of Nusselt number (\(Nu = \frac{hL}{K}\)) which was calculated from computed heat flux on the walls. Fig. 4 shows the variation of Nusselt number Nu with aspect ratio A for different inclination angle θ. This type of heating arrangement for the horizontal cavity (θ=0°) is a hydrodynamically stable case. The heat transfer is purely by conduction and the Nusselt number is 1. But as soon as the inclination θ is increased from 0°, the fluid motion is initiated and the heat transfer is enhanced. The Nusselt number rises above 1. Fig. 4 shows that Nusselt number is unity for all aspect ratios at 0°=0°. For other inclinations, the Nusselt number Nu decreases with increase in aspect ratio A. The amount of deviation of Nusselt number Nu from unity indicates the contribution of natural convection to heat transfer. The results indicate that convection heat transfer contribution is larger at lower aspect ratio.

Fig. 5 shows the variation of Nusselt number Nu with inclination θ for different aspect ratios A. The Nusselt number Nu increases monotonically as inclination θ is increased from 0° to 90° for all aspect ratios. The minimum value of Nu is 1 at θ=0° (horizontal cavity) and the maximum value at θ=90° (vertical cavity). The maximum value of Nu itself is dependent on Aspect ratio A; higher value is obtained for lower aspect ratio cavity. There is no critical inclination for local minima as is found in case of cavity heated from bottom.

Fig. 6 shows comparison of the present results with the correlation due to Arnold et al.[7]. In this correlation, the Nu at any inclination θ is scaled from Nu at 0°=90° by sin(θ) as shown below:

\[Nu = 1 + [Nu(θ=90°) - 1] \sin θ \text{ for } 0° \leq θ \leq 90°\]

The figure shows the values of Nu calculated from the correlation and the present results. Here, the correlation values are based on the value of Nu at 0°=90° obtained from the present CFD predictions. Therefore, the comparison of the results is basically to check the validity of the scaling factor of sin(θ) for inclined cavities in the case of top wall heating. There is very good agreement between the correlation and the present predictions at different inclinations θ and aspect ratios A. It can therefore be concluded that the heat transfer for inclined cavities with top wall heating can be adequately predicted from the knowledge of heat transfer for vertical cavity by simple scaling as given in the correlation of Arnold et al [7]. The plausible reason for success of this scaling is that in inclined cavity the component of g along the walls is g sin(θ) and it is this body force that primarily causes fluid motion in the cavity.
V. CONCLUSIONS

Natural convection flow in a cavity with top wall heating is initiated as soon as the cavity is inclined to the horizontal. The flow pattern is unicellular. The heat transfer increases monotonically from pure conduction at $\theta=0^\circ$ to strong natural convection at $\theta=90^\circ$. The present results confirm that the Nusselt number for inclined cavity can be adequately predicted by scaling the Nusselt number for vertical cavity by $\sin(\theta)$.

REFERENCES


STRESS CONCENTRATION IN ISOTROPIC & ORTHOTROPIC COMPOSITE PLATES WITH CENTER CIRCULAR HOLE SUBJECTED TO TRANSVERSE STATIC LOADING

MOON BANERJEE, N. K. JAIN & S. SANYAL

1, 2 Department of Applied Mechanics, National Institute of Technology, Raipur, Chhattisgarh 492010
3Department of Mechanical Engineering, National Institute of Technology, Raipur, Chhattisgarh, 492010, India

Abstract: The present study brings out the thorough analysis of isotropic and orthotropic fixed rectangular plate with center circular hole under transverse static loading condition. In this paper influence of stress concentration and deflection due to singularity for isotropic and orthotropic composite materials under different parametric conditions is obtained. The effect of thickness-to-width of plate (T/A) and diameter-to-width (D/A) ratio upon stress concentration factor (SCF) for different stresses were studied. An isotropic and one composite material were considered for analysis to determine the variation of SCF with elastic constants. Deflection in transverse direction were calculated and analyzed. Results are presented in graphical form and discussed. Three-dimensional finite element models were created using ANSYS software. Results showed that maximum stress appear near the vicinity of the hole at the upper and lower portions of the plate. The effect of material properties, (E1/E2) on SCF for stresses along x, y and z axis is established thorough this analysis.

Keywords: Composite, stress concentration factor, transverse loading.

1. INTRODUCTION

The usage of composites is increasing in aerospace and other engineering industrial applications, because of their high strength to weight ratios, high stiffness, low density and long fatigue life. As the application of composites to commercial product has increased, so has the need for design aspects for structural components increase. Accurate knowledge of deflections, stresses and stress concentration factors are required for design of such plates with singularities such as circular hole. Any abrupt change in geometry of plate under loading gives rise to stress concentration; as a result, stress distribution is not uniform throughout the cross section.

Rao et al. [1] evaluated the stress around square and rectangular cut-outs in symmetric laminates. It has been analyzed that the maximum stress and its location is mainly influenced by the type of loading. Kumar et al. [2] has studied the post buckling strengths of composite laminate with various shaped cut-outs under in plane shear. Ozen et al. [3] presented the failure loads of mechanical fastened pinned and bolted composite joints with two serial holes. Tsai–Wu failure criterion was used to predict first failure loads by finite element analysis for the geometrical parameters. Ozben et al. [4] compiled FEM analysis of laminated composite plate with rectangular hole and various elastic modulus under transverse loads. Ghezzo et al. [5] performed a numerical and experimental analysis of the interaction between two notches in carbon fibre laminates. Jain and Mittal [6] analyzed the effect of fibre orientation on stress concentration factor in fibrous plate with central circular hole under transverse static loading by using two dimension finite element methods. The numerical analysis of the stress distribution in-plane stress assumption and within the fibrous plate theory framework has been conducted on two symmetric laminates. Mittal and Jain [7] have analyzed the stress concentration and deflection in isotropic, orthotropic and fibrous composite plates with central circular hole subjected to transverse static loading by using two dimensional finite element methods. She and Guo [8] have analyzed the variation of three dimensional stress concentration factors along the wall of elliptic holes in finite thickness plates of isotropic materials subjected to remote tensile stress using finite element method. Gruber et al. [9] developed analytical solution methods for the analysis of stress concentration in fibre reinforced multilayered composites with pin loaded holes. Ukadgaonker and Kakhandik [10] analyzed the stress around an irregular shaped hole for different in-plane loading conditions for an orthotropic fibrous plate. Toubal et al. [11] studied stress concentration in a circular hole in composite plate. Kotousov and Wang [12] have presented analytical solutions for the three dimensional stress distributions around typical stress concentrators in an isotropic plate of arbitrary thickness based on the assumption of a generalized plane strain theory. Troyani et al. [13] have determined the in-plane theoretical stress concentration factors for short rectangular plates with centered circular holes subjected to uniform tension using finite element method. Ukadgaonker and Rao [14] proposed a general solution for stresses around hole in symmetric laminates under in-plane loading by introducing a general form of mapping function and an arbitrary biaxial loading condition to the boundary conditions, and the basic formulation is extended for multilayered plates. Ting et al. [15-16] presented the alternative method to study the stress distributions of the multiple circular or multiple elliptical holes with the rhombic pattern in the infinite domain. Xiwu et al. [17-18] studied a finite composite plate weakened by elliptical holes under different in-plane loading, treated as an anisotropic
Stress Concentration in isotropic & orthotropic composite plates with center circular hole subjected to transverse static loading

In view of the above review, it can be concluded that a detailed analysis of more cases for stress concentration in composite plates with hole subjected to transverse loadings needs to be carried out.

The present work aims to study the effect of T/A and D/A on SCF in isotropic and orthotropic composite (e-glass/epoxy) plates with central circular hole subjected to transverse static loading with all edges fixed and, also the deflection. The effect of T/A and D/A ratio, where T is the plate thickness and A is the plate width and D is the hole diameter on SCF for normal stresses in X, Y, Z directions (σx, σy, σz), shear stress (τxy) and deflection in transverse direction (Uz) is investigated by using three dimensional finite element analysis. The deflection in z-direction for plate with hole (Uz) of different materials under transverse loading is compared with deflection in transverse direction in plate without hole (Uz*). Results are obtained for one isotropic and composite material to find out the variation of SCF for different material parameters.

II. FORMULATION OF PROBLEM

The model of fixed plate of dimension (0.2 m X 0.1 m) having thickness (T) with a central circular hole of diameter (D) under uniformly distributed loading of P (N) in transverse direction (Fig. 1) is taken for analysis. The plate is fixed at all edges.

The material properties [19] of isotropic plate are as: [E, μ]: [39 GPa, 0.3] and, of orthotropic composite plate are as: [(E1, E2, E3, G12, G23, G31, μ12, μ23, μ31): [39, 8.6, 8.6, 3.8, 3.8, 3.8 GPa, 0.28, 0.28, 0.28]. Here, E, G and μ represent modulus of elasticity, modulus of rigidity and Poisson’s ratio.

III. ANALYSIS

Finite element method was chosen for analysis of the model. The model was meshed using a 3-D solid element, Solid 186 with three degrees of freedom and 60 nodes per element in ANSYS. Typical mesh of the plate using the above element has been shown in Figure. 2. Mapped meshing is used so that more elements employed near the hole boundary. Due to the symmetric nature of different models investigated, it was necessary to discretize the quadrant plate for finite element analysis. Main task in finite element analysis is selection of suitable elements. Numbers of checks and convergence test are made for selection of suitable elements from different available elements and to decide the element length. Element length is selected as 0.002 m after running the convergence tests. The example of the discretized three dimensional finite element model, used in study is shown in Fig. 2.

IV. RESULT AND DISCUSSION

Models generated are analyzed and results thus obtained for material combinations are presented in graphs. Variation of SCF (for σx, σy, σz, τxy), deflection versus D/A ratios for all edges fixed rectangular plate with central circular hole loaded transversely for two different materials were presented in below figures. Results are discussed case by case further. Fig. 3 shows variation of SCF (σx) versus D/A ratio for isotropic and e-glass/epoxy material.

Following observations can be made from the analysis represented in Fig.3. Both materials considered are following a similar behavior with continuous decrease in SCF (σx) with corresponding increase in D/A and T/A ratio respectively. Isotropic and e-glass/epoxy material shows their maximum SCF value of 1.02, 1.35 at T/A=0.10 and D/A=0.1 respectively, whereas minimum SCF value of 0.89, 0.87, were obtained at T/A=0.01 and D/A=0.5 ratio respectively.
Fig. 4 shows variation of SCF ($\sigma_x$) versus D/A ratio for isotropic and e-glass/epoxy material.

Isotropic material show increase in SCF ($\sigma_x$) with corresponding increase in D/A ratio and decrease in SCF with continuous increase in T/A ratio. Isotropic material show maximum SCF ($\sigma_x$) of 1.07 at T/A=0.01 and D/A=0.1 ratio, whereas minimum SCF of 0.98 is attained at T/A=0.10 and D/A=0.5 ratios. E-glass/epoxy follows the opposite behavior for increase in SCF ($\sigma_x$) with corresponding increase in T/A and D/A ratios respectively. Maximum SCF ($\sigma_x$) of 0.82 is obtained for T/A=0.10 and D/A=0.5 parameters, whereas minimum SCF of 0.70 is obtained at T/A=0.01 and D/A=0.1 respectively.

Fig. 5 shows variation of SCF ($\sigma_z$) versus D/A ratio for isotropic and e-glass/epoxy material. For isotropic material SCF ($\sigma_z$) decrease with continuous increase in D/A ratio and increase in SCF with corresponding increase in T/A ratio up to T/A=0.05 and then it decreases. E-glass/Epoyxy shows continuous increase in SCF ($\sigma_z$) with corresponding increase in both D/A and T/A ratios. Isotropic material attains its maximum SCF ($\sigma_z$) value as 1.09 at T/A=0.05 and D/A=0.1 ratio, whereas minimum SCF value of 0.89 is obtained at D/A=0.5 and T/A=0.01 parameters. E-glass/Epoyxy obtains its maximum SCF value as 0.67 at T/A=0.10 and D/A=0.5, whereas minimum SCF value of 0.48 is attained at T/A=0.01 and D/A=0.1 ratios respectively.

Fig. 6 shows variation of SCF ($\tau_{xy}$) versus D/A ratio for isotropic and e-glass/epoxy material. Following observations can be made from the analysis represented in Fig. 6. Isotropic material follows decrease in SCF ($\tau_{xy}$) with corresponding
increase in D/A ratio and increase in SCF ($\tau_{xy}$) with continuous increase in T/A ratio respectively. Isotropic material show maximum SCF ($\tau_{xy}$) of 2.55 at T/A=0.10 and D/A=0.5 respectively, whereas minimum SCF of 1.33 is attained at T/A=0.01 and D/A=0.5 ratios. E-glass/epoxy follows the decrease in SCF ($\tau_{xy}$) with corresponding increase in D/A ratio. E-glass/epoxy have its maximum SCF of 3.30 obtained for T/A=0.10 and D/A=0.1, whereas minimum SCF of 2.60 is obtained at T/A=0.01 and D/A=0.1 respectively.

Fig. 7 shows variation of $U_z/U^*_z$ versus D/A ratio for isotropic and e-glass/epoxy material.

It has been observed that $U_z/U^*_z$ for isotropic material increases initially with increase in D/A ratio up to 0.2 and after D/A=0.2, $U_z/U^*_z$ decrease with corresponding increase in D/A ratios. $U_z/U^*_z$ for isotropic material show maximum SCF value of 1.09 at T/A=0.01 and D/A=0.2 ratios respectively, whereas minimum $U_z/U^*_z$ of 0.89 is obtained at T/A=0.10 and D/A=0.5 parameters. E-glass/epoxy follows opposite trend as compared to isotropic material with increase in $U_z/U^*_z$ for corresponding increase in D/A and T/A parameters. Maximum $U_z/U^*_z$ value of e-glass/epoxy is 0.64 obtained for T/A=0.10 and D/A=0.5 ratios, whereas minimum value of 0.56 attained at T/A=0.01 and D/A=0.1 ratios respectively.
V. CONCLUSION

Detailed investigations on the stress fields and the relationships between three-dimensional stress concentration factor (SCF) for isotropic and composite plate subjected to transverse loading were conducted using 3D finite element method:-

- The influence of D/A and T/A parameters shows a substantial role for all stresses and ratios of $U_x/U_y$ for both materials considered.
- Among all the stresses, SCF (for $\tau_{xy}$) is seen maximum on the hole boundary along the width direction of the plate, whereas remaining stresses were maximum on support edges of the plate.
- Higher $E_x/E_y$ & $E_y/G_{xy}$ ratios, prominently effect the values of deflection and SCF (for $\sigma_x$, $\sigma_y$, $\tau_{xy}$) respectively. Thus both material graph behave differently for all stresses and deflection because of large difference in their material parameters.

Additionally this can also be concluded that the results obtained are in line with other similar works.

ACKNOWLEDGMENT

Authors acknowledge the authorities of N.I.T, Raipur for permitting the above work to be carried out in the Department of Applied Mechanics

REFERENCES


CONTAMINANT CONTROL IN INTENSIVE CARE UNIT (ICU) USING CFD MODELING

TIKENDRA NATH VERMA & S.L.SINHA,

Department of Mechanical, National Institute of Technology, Raipur, Chhattisgarh 492010

Abstract:-Computational fluid dynamic (CFD) analysis is used to simulate and compare the removal of microbes using a number of different ventilation systems in hospitals. The primary objective of ventilation system design in hospital is to place the patient at no risk of infection while hospitalization. Normally hospitals are considered to be clean and free from pathogens which are actually not true. Due to the complex environment of hospital, the effective ventilation for comfort of patients & control of infections must be given highest priority. Intensive care represents the highest level of continuing patient care and treatment. Therefore a turbulent airflow study has been performed in Intensive Care Unit (ICU) of hospital. The present investigation stresses preventing airborne infections, protecting the doctor and other patient in ICU, using Computational Fluid Dynamics (CFD) software FLUENT. In which, Navier Stokes and energy equations in three-dimensional co-ordinates have been solved by control volume method. The SIMPLE algorithms are used to solve these equations. Steady state, k-ε turbulence model and incompressible flow of a constant property fluid have been considered. The tracking of massless contaminated particle (infection) has also been carried out by simulation. It is observed that remote pocket of the room where air circulation is not proper, is not healthy for the patients as well as doctor. Therefore suitable ventilation arrangement must be provided for healthy environment in the hospital.

Keywords- Air flow, CFD, ICU, Micro-organism, Ventilation.

1. INTRODUCTION

Isolation precaution is an important strategy in the practice of infection control. The spread of some infections can be impeded if infected patients are segregated from those who are not yet infected. Although there is no single study showing the effectiveness of isolation, there are many reports documenting the efficacy of the various components of isolation, including use of private rooms and protective equipment’s such as masks, gloves and gowns.

Towards the end of the 19th century, there were recommendations for patients with infectious diseases to be placed in separate facilities, which ultimately became known as infectious diseases hospitals. However, in the early 1950s, many of these infectious diseases hospitals closed and the patients were moved to general hospitals. The need for proper isolation of infections in the context of these general hospitals thus became an important issue. Airborne transmission occurs by dissemination of droplet nuclei over long distance from infectious patients. Infectious agents that may be dispersed over long distances by air currents and infect other susceptible individuals include Mycobacterium (tuberculosis), rubeola virus (measles) and Varicella-zoster virus (chickenpox).

The biological quality of air in hospital environments is of particular concern as patient may serve as a source of pathogenic microorganism to staff and hospital visitor in addition to other admitted patients. The most important source of airborne pathogens inside the hospital is infected patient. The airborne transmission of pathogens occurs when it is transferred from an infected patient to other people. The present study includes the simulation of such cases to control the infection in the surrounding areas in the hospital. The need of precise determination of airflow pattern and temperature distribution in a room was realized at first by air conditioning engineers so as to provide comfort condition of temperature and air velocity throughout the occupied zone. In modern era, people spends about 90% of the time in indoor environment such as home, offices, factories, transport vehicles, recreational buildings, hospital etc. In hospitals since more than 8000 chemical species have been identified in the indoor environment.

CFD was initiated around 1930. The concept of turbulence was introduced into calculation of room airflow after 1970. Helmis et al. [1] have presented an experimental and theoretical study on assessing the status of air quality in a dentistry clinic with respect to chemical pollutants and identifying the indoor sources associated with dental activities. Different schemes of natural ventilations were explored to examine their effects on the indoor comfort conditions for the occupants in terms of air renewal. Huang & Tsao [2] studied ventilation conditions, impact dispersion of pathogenic nuclei in an AIIR (Airborne infection isolation room) by investigating the airflow conditions and impacting dispersion of infectious agents in it. The simulations were performed on a fine tetrahedral mesh with approximately 1.3×10^6 cells in AIIR. Rui et al. [3] have studied the airborne transmission of bacteria in two operating rooms during two surgeries, a stitching of fractured mandible and a joint replacement. The results showed that improving airflow pattern could reduce particle deposition on critical surfaces. Jayaraman et al. [4] reported a CFD study of containment of airborne hazardous materials in a ventilated room containing a downdraft table with the consideration of arrangement of ventilation configurations. Li et al. [5] showed the concern for lack of protection within an AIIR, it is important to
develop an understanding of air and contaminant transport in the room. Balaras et al. [6] presented an overview of general design for acceptable indoor conditions related to HVAC systems in hospital operating rooms. Lewis et al. [7] studied the influence of room air distribution on the infection rate in an operating room and concluded that an optimal air distribution plays an important role in maintaining the proper environmental condition within a surgical room. Memarzadeh et al. [8] proposed a methodology for minimizing contamination risk from airborne organisms in hospital isolation rooms. The results show that the number of particles deposited on surfaces and vented out is greater in magnitude than the number killed by ultraviolet (UV) light, suggesting that ventilation plays an important role in controlling the contaminant level.

II. ASSUMPTION USED IN MODELING

Numerical modeling for the airflow is based on the following assumptions:
1. Aerodynamic blockage (obstacle) due to presence of human block available in the room has been considered.
2. Presence of heat and pollution sources has been considered.
3. Physical properties such as density, conductivity etc. is assumed to be constant.
4. The flow is considered to be steady, turbulent and incompressible under Boussinesq’s approximation.

III. GOVERNING EQUATIONS

Computational fluid dynamics (CFD) models are used to predict the air velocity, turbulence level & air temperature. The prediction of airflow in an ICU, the flow equation must account for turbulence and buoyancy. Conservation equations for mass, momentum & energy can be established for each cell.

Three dimensional General Transport equations for turbulent flow are given below:

$$\frac{\partial (\rho \phi)}{\partial t} + \frac{\partial (\rho \phi u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \Gamma_\phi \frac{\partial \phi}{\partial x_i} \right) + S_\phi$$

Table 2 shows the details of variables $\phi$, $\Gamma_\phi$ and $S_\phi$ for various conservation equations. The inlet velocity $U_i$ and inlet opening $W_i$ are taken as characteristic velocity and length respectively. The skin temperature of human body ($T_0$) is 37°C. The difference between inlet air temperature and maximum wall temperature ($\Delta T$) has been used for non-dimensionlization of temperature.

IV. THE LAYOUT OF COMPUTER SIMULATION MODEL

Figure 1 shows the isometric view of ICU. The overall dimensions of ICU have been considered as 4.0 m ×3.0 m ×6.0 m in X, Y and Z directions, respectively. In the numerical grid, the doctor and patient were treated as rectangular solid boxes of size 0.3 m ×1.7m ×0.5 m and 0.3 m ×1.5m ×0.5 m respectively. The walls, Floor and ceiling of the room were all well insulated. The supply air opening of size 0.6m ×0.4m has been placed on south wall 2.3m above the floor in different position i.e. 0.6m, 1.7m, 2.8m far from west wall & the exhaust outlet of same size as inlet has been placed on north wall. The inlet velocity considered is 0.5 m/sec, air change rate per hour (ACH) of 6 and inlet temperature is 20°C. The temperatures of different walls on east, west, north, south, and ceiling have been taken as 31°C, 25°C, 17°C, 28°C & 48°C respectively from ISHRAE handbook for Raipur region [9].

The following three cases of fixed outlet position and different inlet positions have been simulated as shown in Table 1:

<table>
<thead>
<tr>
<th>Case No.</th>
<th>Position of Inlet from west wall</th>
<th>Position of Outlet</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>0.6m</td>
<td>0.3m above the floor &amp; 1.7m infront of west wall</td>
<td>Inlet height from the floor is 2.3 m in all the cases.</td>
</tr>
<tr>
<td>2.</td>
<td>1.7m</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.</td>
<td>2.8m</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The numerical model solves mass, momentum and energy equations. The Standard k-turbulence model has been used. The velocity-pressure coupling is solved by means of the SIMPLE [Semi-Implicit Method for Pressure-Linked Equations] method. The computational domain has $3.8 \times 10^5$ cells. No-slip boundary conditions were applied on all walls. The first-order upwind scheme was used for discretizing the convection terms. The convergence criteria for the air properties (pressure, energy, k and $\varepsilon$) were assumed to have been met when the iteration residuals have reduced to $10^{-5}$.
V. RESULT & DISCUSSION

In ICU, presence of two patients, one doctor and light have been considered as heat source. The simulated results have been plotted on plane $x=1m$, $2m$, & $3m$ as shown in figure 2.

Figures 3(a-d) show the temperature contour (TC) of ICU at planes $x=1m$, $2m$ and the path travelled by contaminated particle produced from the mouth of the patient 1 & patient 2. In this case, the inlet is 0.6m away from west wall. Figure 3(a) clearly shows that the cold air is entering through the inlet and moves almost touching the ceiling and falling before 2.5m and spreads in the room. Due to these movements the temperature in the room is observed as non-uniform.

In this case, the local mean age of the contaminated particle is approximately 20 minutes.

Figure 3(b) the temperature is observed to be high near the ceiling due to available light load. The temperature in the occupied zone is observed to be uniform. The corner part of this plane is observed to be colder than the occupied zone. It may be due to the less circulation of hot air coming from the light source.

Figure 3(c) shows the path travelled by contaminated particle produced from the mouth of first patient. In this case, it is assumed that no contamination is produced from patient 2. The contaminated particle starts from the mouth of patient 1 and moves in a tortuous path and leaves through outlet.

Figure 3(d) show the path travelled by contaminated particle produced from the mouth of second patient. In this case, it is assumed that no contamination is produced from the first patient. The contaminated particle starts from the mouth of patient 2 and leaves through the outlet without affecting the other patient. Here the local mean age of the contaminated particle is quite less as compared to Fig 3(c). It indicates healthy environment in the room.

In this case, the local mean age of the contaminated particle is approximately 20 minutes.

Figure 3(d) show the path travelled by contaminated particle produced from the mouth of second patient. In this case, it is assumed that no contamination is produced from the first patient. The contaminated particle starts from the mouth of patient 2 and leaves through the outlet without affecting the other patient. Here the local mean age of the contaminated particle is quite less as compared to Fig 3(c). It indicates healthy environment in the room.
Contaminant control in intensive care unit (ICU) using CFD modeling

from the mouth of the patient 1 & patient 2. In this case, the inlet is 1.7 m infront from west wall i.e. at the middle of the south wall. Figure 4(a) clearly shows that the cold air is entering through the inlet and moves straight almost touching the ceiling and falling before 3.6m and spreads in the room. Due to these movements the temperature in the room is observed to be non-uniform.

Figure 4(b) shows the path travelled by contaminated particle produced from the mouth of first patient. It is assumed that no contamination is produced from patient 2. The particle shows the little tortuous path but it is not affecting the other patient and doctor.

Figure 4(c) shows the path travelled by contaminated particle produced from the mouth of second patient. It is assumed that no contamination is produced from patient 1. The particle follows a little tortuous path and leaves through the outlet. Here the local mean age of the contaminated particle is quite less as compared to Fig 4(b). It indicates healthy environment in the room. Thus, it is not affecting the health of the patient and doctor.

Figures 5(a-d) show the temperature contour of ICU at plane x=2m, 3m and the path travelled by contaminated particle produced from the mouth of the patient 1 & patient 2. In this case, the position of inlet is 2.7 m far from west wall. Figure 5(a) shows that the cold air is entering the inlet & moves almost touching the ceiling & falling before 2.5m and spreads in the room.

Figure 5(b) shows that the temperature is observed to be high near the ceiling due to available light load in ICU. The temperature in the occupied zone is observed to be uniform. The corner part of this plane is observed to be colder than the occupied zone. It may be due to the less circulation of hot air coming from the light source.
prominent when the ICU is provided inlet at 1.7m away from west wall, here local mean age of contaminated particle is less and it goes out easily through outlet without affecting the health of patient and doctor. Hence the ventilation provided in the second case is the best option as far as the healthy environment is concerned.

REFERENCES


# Table 2. Notations for Governing Equations in Cartesian Co-ordinates for Turbulent Flow

<table>
<thead>
<tr>
<th>Equation</th>
<th>$\phi$</th>
<th>$\Gamma_\phi$</th>
<th>$S_\phi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>u-momentum</td>
<td>$\frac{1}{Re} \frac{\mu_t}{Re}$</td>
<td>$-\frac{\partial}{\partial x} \left( \Gamma_\phi \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \Gamma_\phi \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial z} \left( \Gamma_\phi \frac{\partial u}{\partial z} \right)$</td>
<td></td>
</tr>
<tr>
<td>v-momentum</td>
<td>$\frac{1}{Re} \frac{\mu_t}{Re}$</td>
<td>$-\frac{\partial}{\partial y} \left( \Gamma_\phi \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial x} \left( \Gamma_\phi \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial z} \left( \Gamma_\phi \frac{\partial v}{\partial z} \right)$</td>
<td></td>
</tr>
<tr>
<td>w-momentum</td>
<td>$\frac{1}{Re} \frac{\mu_t}{Re}$</td>
<td>$-\frac{\partial}{\partial z} \left( \Gamma_\phi \frac{\partial w}{\partial z} \right) + \frac{\partial}{\partial y} \left( \Gamma_\phi \frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial x} \left( \Gamma_\phi \frac{\partial w}{\partial x} \right)$</td>
<td></td>
</tr>
<tr>
<td>Energy</td>
<td>$\frac{1}{ReRe} \frac{\mu_t}{ReRe}$</td>
<td>$0$</td>
<td></td>
</tr>
<tr>
<td>Turbulence Kinetic energy</td>
<td>$\frac{1}{Re} \frac{\mu_t}{ReRe}$</td>
<td>$\frac{\epsilon}{Re} \left( C_1 f_1 G_{\mu t} - C_2 f_2 G_{\mu t} + C_3 G_{\mu t} \right)$</td>
<td></td>
</tr>
<tr>
<td>Heat dissipation rate</td>
<td>$\frac{1}{Re} \frac{\mu_t}{ReRe}$</td>
<td>$\frac{\epsilon}{Re} \left( C_1 f_1 G_{\mu t} - C_2 f_2 G_{\mu t} + C_3 G_{\mu t} \right)$</td>
<td></td>
</tr>
</tbody>
</table>
COMPARISON OF VARIOUS NUMERICAL DIFFERENCING SCHEMES IN PREDICTING NON-NEWTONIAN TRANSITION FLOW THROUGH AN ECCENTRIC ANNULUS WITH INNER CYLINDER IN ROTATION

SATISH KUMAR DEWANGAN¹ & S. L. SINHA²

¹Assistant Professor, Applied Mechanics Department, National Institute of Technology, Raipur, C.G. (INDIA), ²Associate Professor, Mechanical Engineering Department, National Institute of Technology, Raipur, C.G. (INDIA),

Abstract: Flow through annulus is a widely solved problem in fluid mechanics because of its practical applicability in many areas. In oil well drilling, cuttings generated are of non-Newtonian nature which flows through an eccentric annulus with inner cylinder in rotation, considering the real drilling situations. In the present work, a comparison have been done amongst three differencing schemes, the second order upwind, the Power law scheme and QUICK scheme used in computational fluid dynamics solution algorithms, so as to find the best amongst them to solve transition flow for this case as well as in general. A three dimensional orthogonal hexahedral mesh with suitable boundary conditions & input parameters was taken as computational domain for eccentric annulus. This was solved with standard k-ω turbulence model and SIMPLE algorithm. Results were validated against the published experimental work of J. M. Nouri, et. al [1].

Radial velocity, axial velocity and tangential velocity of fluid were plotted along chosen planes and contours of molecular viscosity as well as turbulence kinetic energy were observed for comparison amongst the solutions obtained by three differencing schemes. Although in most of the cases close agreement have been observed between computational data, but as far as prediction of radial velocity is concerned there was surprising difference amongst the three schemes.

Keywords: Non-Newtonian Flow; Transition Flow; Eccentric Annulus.

1. INTRODUCTION

Flow through annulus is a widely solved problem. Chemical process & petroleum industries, Pipeline engineering, Power plants, Biomedical engineering applications, Micro scale fluid dynamics studies, Food processing industries, geothermal flows, extrusion of molten plastic etc encounter many applicable situations of flow through such geometrical situation of concentric annulus or eccentric annulus. A large variety of fluids and industrial applications has been a major motivation for research in annular flow with varying degrees of complexity. An extensive bibliographic list of work on annular flows has been presented by Escudier et al. [2]. Of concern to this work are mainly previous investigations with viscoelastic fluids in concentric annuli under laminar flow conditions.

Usual situation occurring in the case of oil well and gas well drilling mud flow is either transition or turbulent situation. In the work of J.M. Nouri [1] et al. three velocity components (axial, radial and tangential) of a Newtonian and a weakly elastic shear-thinning non-Newtonian fluid have been measured in an annulus with an eccentricity of 0.5, a diameter ratio of 0.5, and an inner cylinder rotation of 300 rpm. The results show that the rotation had similar effects on the Newtonian and non-Newtonian fluids, with a more uniform axial flow across the annulus and the maximum tangential velocities in the narrowest gap in both cases. The turbulence intensities in the region of widest gap were uninfluenced by rotation, increased in the Newtonian fluid, and decreased in the non-Newtonian fluid in the region of the smallest gap. D.O.A. Cruz et. al. [3] has obtained analytical solution of helical flow of fluids in concentric annuli due to inner cylinder rotation as well for Poiseuille flow in a channel skewed by the movement of one plate in span wise direction, which constitutes a simpler solution for helical flow in the limit of very thin annuli. Expressions are derived for the radial variation of the axial and tangential velocities, as well as for the three shear stresses and the two normal stresses using non dimensional no as Reynolds No and Taylor No etc. I. A. Frigaard et. al. [4] has worked in predicting the rheological properties that are necessary to prevent the annular plug fluid from flowing under the action of buoyancy, or indeed to predict how far the plug material may flow for given rheological properties for annular fluid flow in oil wellbore construction. Mathematically, these flows were modelled using a Hele-Shaw approximation of the narrow annulus. V. C. Kelessidis (SPE) et. al. [5] has presented a critical review of the state-of-art modelling for efficient cutting transport during Coiled-tube drilling, and presented the critical parameters like pump rate, well dimension, fluid sizes, solid loading and hole inclination etc affecting efficient cutting transport. They set up a laboratory system also. M. P. Escudier et. al. [6] reports experimental data for fully developed laminar flow of a shear-thinning liquid through both a concentric and an 80% eccentric annulus with and without centre body rotation. The working fluid was an aqueous solution of 0.1% xanthan gum and 0.1% carboxy-methyl cellulose. A. A. Gavrillovet. al. [10] proposed a numerical...
algorithm for simulating steady laminar flows of an incompressible fluid in annular channels with eccentricity and rotation of the inner cylinder. The algorithm enabled description of this class of flows for wide ranges of the annular channel and flow parameters. For a series of flows in an annular clearance, these numerical results were compared with the available analytic solutions and experimental data. The simulated data agree well with the available experimental, analytical, and numerical solutions. Sang-Mok Han et al. [11] investigated hydraulic transport characteristics of a solid-liquid mixture flowing vertically upward where solid particles are carried by non-Newtonian fluids in a slim hole concentric annulus with rotating inner cylinder. Solid volumetric concentration and pressure drops were measured for the various parameters such as inclined annulus, flow rate, and rotational speed of inner cylinder. Aqueous solution of sodium carboxymethyl cellulose 0.2 ~ 0.4% (CMC) and 5% bentonite solutions were taken for non-Newtonian fluid one by one. For both CMC and bentonite solutions, the higher the concentration of the solid particles are, the larger the pressure drops become. Wang Zhiyuan [12] established the basic hydrodynamic models, including mass, momentum, and energy conservation equations for annular flow with gas hydrate phase transition during gas kick for deep water drilling. They investigated the behavior of annular multiphase flow with hydrate phase transition by analyzing the hydrate-forming region, the gas fraction in the fluid flowing in the annulus, pit gain, bottom hole pressure, and shut-in casing pressure. Results show that it is possible to move the hydrate-forming region away from sea floor by increasing the circulation rate. The decrease in gas volume fraction in the annulus due to hydrate formation reduces pit gain, which can delay the detection of well kick and increase the risk of hydrate plugging in lines. Caution is needed when a well is monitored for gas kick at a relatively low gas production rate, because the possibility of hydrate presence is much greater than that at a relatively high production rate. The shut-in casing pressure cannot reflect the gas kick due to hydrate formation, which increases with time. Young-Ju Kim et al. [13] did an experimental investigation concerning the characteristics of vortex flow in a concentric annulus with a diameter ratio of 0.52, whose outer cylinder is stationary and inner one is rotating. Pressure losses and skin friction coefficients have been measured for fully developed flows of water and of 0.4~o aqueous solution of sodium carboxy-methyl cellulose (CMC), respectively, when the inner cylinder rotates at the speed of 0~600 rpm. Also, the visualization of vortex flows has been performed to observe the unstable waves. E. V. Podryabinkinet al. [14] presents results of numerical modelling for analysis of the moment and forces exerted on an eccentrically positioned rotating inner cylinder due to the annular flow between two cylinders with parallel axes. Laminar stationary fully developed flows of Newtonian and power law fluid flows are considered. An impact of annulus geometry, flow regime, and fluid characteristics are studied. The study indicates that the moment exerted on the inner cylinder increases monotonically with the eccentricity. Forces acting on the inner cylinder include pressure and viscous friction. The pressure forces provide a predominant contribution. When eccentricity does not exceed a certain critical value, the radial force pushes the inner cylinder to the channel wall. When eccentricity is large enough, the radial force reverses its sign, and the inner cylinder is pushed away from the outer wall. Circumferential component of the force has always the same direction and induces precession of the inner cylinder. Sang-mok Han [15] experimentally studied solid-liquid mixture upward hydraulic transport of solid particles in vertical and inclined annuli with rotating inner cylinder. Effect of annulus inclination and drill pipe rotation on the carrying capacity of drilling fluid, particle rising velocity, and pressure drop in the slim hole annulus have been measured for fully developed flows of water and of aqueous solutions of sodium carboxy methyl cellulose (CMC) and bentonite, respectively. For higher particle feed concentration, the hydraulic pressure drop of mixture flow increases due to the friction between the wall and solids or among solids.

In these researches, depending upon the type and nature of the flow different solution methodology has been adopted. Usually turbulent & transitional non-Newtonian flows are often encountered in the oil and gas industry. These fluids are used in the drilling of oil wells to transport the cuttings to the surface, and to keep solids in suspension during stationary periods. In directional drilling, an eccentric annulus is often used, there is a tendency for the cuttings to accumulate in the narrowest gap where the velocity is lowest. The cutting generated is of Non-Newtonian nature. And although the cutting is a directional process, still along with the axial annular flow involves lateral rotational effects because of the rotation of inner cylindrical portion of the annulus. Such combination of axial annulus flow along with the lateral rotation brings the fluid in that zone to be in transition or in turbulence state. This tends to suppress such accumulation of cuttings. Thus it is very much important to maintain transition or turbulent flow situation. This, in turn, requires knowledge of the velocity profiles and other flow characteristics in the annulus as essential in the proper design and operation of the drills.

In computational solution procedure, while solving such problems selection of proper turbulence model, selection of proper pressure – velocity coupling resolution method and type of discretization schemes for flow are very essential. There are various...
upwind schemes to capture the flow effects depending on the dominance of flow direction & magnitude.

When flow of fluid plays a significant role, the convection effects of flow must be taken into account while solving these conservation equations for boundary conditions given in a problem under consideration. This requires selection of right type of differencing scheme for the flow properties & other variables (i.e. momentum terms, pressure, turbulence kinetic energy etc) involved. The various differencing schemes for predicting convective effects of flow are; upwind scheme, Power law schemes and QUICK scheme etc., (in the increasing order of effectiveness). Upwind differencing scheme is simple & thus can be extended to multidimensional problems but has a drawback that it produces erroneous results when the flow direction is not aligned with grid lines. The upwind differencing scheme causes the distributions of the transported properties to become smeared & cause false diffusion. Power law scheme is fully conservative, unconditionally bounded is highly stable & produces realistic solutions. This scheme is very useful in predicting practical flows QUICK scheme has very small false diffusion problem and the solution achieved with coarse grids are often considerably more accurate than those of former schemes. Thus effectiveness of these differencing schemes in capturing the convection effects is judged by how far they are able to satisfy the three qualities namely Conservativeness, Bounded-ness and Transportiveness. H. K. Versteeg et al. [7].

The objective of the present study is to investigate the flow prediction effectiveness of second order upwind and power law differencing schemes, in respect to QUICK schemes for case of eccentric annulus (Refer Table 1 for model details) non-Newtonian flow with inner cylinder rotation. Before it, the results of QUICK scheme have been verified by comparing them with that of experimental data for the case considered. Three dimensional and two dimensional (mid-plane of the annulus normal to the annulus axis) representation of the geometry is shown in Figure 1 & Figure 2 respectively.

### Table 1: Model Details: Reference [8]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder Diameter (Dc)</td>
<td>20 mm</td>
</tr>
<tr>
<td>Outer cylinder Diameter (D0)</td>
<td>40.3 mm</td>
</tr>
<tr>
<td>Max hydraulic mean diameter (Dh)</td>
<td>20.3 mm</td>
</tr>
<tr>
<td>Maximum Eccentricity (emax)</td>
<td>5.15 mm</td>
</tr>
<tr>
<td>Axial dimension for the cylinders</td>
<td>10 mm</td>
</tr>
<tr>
<td>Inner cylinder is in clockwise rotation</td>
<td>300 rpm.</td>
</tr>
<tr>
<td>Outer cylinder is fixed.</td>
<td></td>
</tr>
<tr>
<td>Centre of the inner cylinder is taken as reference centre for measurement of the eccentricity.</td>
<td></td>
</tr>
</tbody>
</table>

Flow is considered to be three dimensional, incompressible, steady & transition with Bulk axial Reynolds no (Re) equal to 9000. The flow direction is along positive z-axis, through the eccentric annulus of flow between the cylinders. Axial mass flow rate (m) & axial bulk velocity (U) corresponding to the chosen Reynolds no was 2.615 kg/s and 2.72 m/s, which were calculated using the following relations;

- Axial Bulk velocity; \( U = \frac{(R_e \times D_H \times \rho)}{\mu_{wall}} \)
- Axial mass flow rate; \( m = U \times A \times \rho \)

Where \( A = \frac{\pi}{4} (D_0^2 - D_i^2) \) and \( \mu_{wall} = 6 \times 10^{-3} \) measured experimentally by J. M. Nouriet. et al. [1].

### Table 2: Non-Newtonian power law parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Power law index (n)</td>
<td>0.75</td>
</tr>
<tr>
<td>consistency index (K)</td>
<td>0.044</td>
</tr>
<tr>
<td>Minimum viscosity Limit (( \mu_{min} ))</td>
<td>0.0001 kg/m-s</td>
</tr>
<tr>
<td>Maximum Viscosity Limit (( \mu_{max} ))</td>
<td>1000 kg/m-s</td>
</tr>
<tr>
<td>Reference Temperature</td>
<td>310 K</td>
</tr>
</tbody>
</table>

The values of these non-Newtonian power law parameters for test fluid used in this study were obtained from the dataprescribed by J. M. Nouriet. et al. [1].

The numerical formulation is based on the finite volume method as implemented in ANSYS-FLUENT 12. SIMPLE algorithm has been applied for dealing with pressure velocity coupling. The present study employs standard k-omega turbulence model with transition flow option & shear flow corrections.
II. MATHEMATICAL AND NUMERICAL FORMULATION

In the present work, flow has been modelled as three dimensional, incompressible, transition flow of non-Newtonian fluid. ANSYS-FLUENT solves the problem using conservation equations as guideline. Flow of fluid is governed by the Navier–Stokes equation and continuity equations. The coordinate-free time averaged form of the Navier-Stokes equations (K. Muralidhar et. al. [9]) is being given below;

\[ \rho \left( \frac{\partial \bar{U}}{\partial t} + \bar{U} \cdot \nabla \bar{U} \right) = -\nabla p + \mu \nabla^2 \bar{U} + \nabla \cdot \mathbf{T} \]

Time averaged velocity components of turbulent flow satisfy the same Navier-Stokes equation as for laminar flow, provided the laminar stresses are increased by additional stress known as apparent stresses of turbulent flow or Reynolds stresses. These are given by symmetric stress tensor as below;

\[ \mathbf{T} = \left[ \begin{array}{ccc} \frac{\partial \bar{U}_r}{\partial r} & \frac{\partial \bar{U}_r}{\partial z} & \frac{\partial \bar{U}_r}{\partial r} \\ \frac{\partial \bar{U}_z}{\partial z} & \frac{\partial \bar{U}_z}{\partial z} & \frac{\partial \bar{U}_z}{\partial r} \\ \frac{\partial \bar{U}_r}{\partial r} & \frac{\partial \bar{U}_z}{\partial r} & \frac{\partial \bar{U}_z}{\partial r} \end{array} \right] 

Details of velocity vector \( \bar{U} \) and Del operators in radial coordinates are as below;

\[ \bar{U} = \bar{U}_r \bar{e}_x + \bar{U}_z \bar{e}_z + \bar{U}_a \bar{e}_x \]

\[ \nabla^2 \bar{U} = \nabla (\nabla \bar{U} \cdot \nabla ) + \frac{1}{r^2} \frac{\partial }{\partial r} \left( r^2 \frac{\partial \bar{U}}{\partial r} \right) + \frac{1}{r} \frac{\partial }{\partial \theta} \left( r \frac{\partial \bar{U}}{\partial \theta} \right) \]

The second term is:

\[ \frac{\partial^2 \bar{U}_r}{\partial r^2} - \bar{U}_r = - \frac{2}{r^2} \frac{\partial \bar{U}_r}{\partial r} \]

\[ + \left( \frac{\partial^2 \bar{U}_r}{\partial r^2} + 2 \frac{\partial \bar{U}_r}{\partial r} - \bar{U}_r \right) \bar{e}_x \]

The nonlinear acceleration term would be;

\[ \bar{U} \cdot \nabla \bar{U} = \sum_i \left( \frac{\partial \bar{U}_i}{\partial \theta} \right) \bar{U}_i \bar{e}_x + \frac{1}{r} \frac{\partial }{\partial r} \left( r \frac{\partial \bar{U}_i}{\partial r} \right) \bar{e}_r + \frac{\partial \bar{U}_i}{\partial r} \bar{e}_r \]

The continuity equation for the incompressible fluid is given as under;

\[ \nabla \cdot \bar{U} = 0 \]

Writing the Navier–Stokes equations in this form, allows the flexibility to use arbitrary non-Newtonian fluid model. Energy Conservation equation will not play any role since thermal parameters are not varying.

ANSYS-FLUENT provides four options for modelling non-Newtonian flows: (a) power law model, (b) Carreau model for pseudo-plastics, (c) Cross model and (d) Herschel-Bulkley model for Bingham plastics. The test fluid is of non-Newtonian type. It has been described by power law model and represented by (when temperature is not involved in the case under study) the following equation;

\[ \mu = K \gamma^{n-1} \]

Here, \( K \) is the measure of the average viscosity of the fluid (the consistency index); \( n \) is a measure of the deviation of the fluid from Newtonian state (the power law index). If viscosity computed from the power law crosses these maximum or minimum limits then extreme value of that side will be used instead for calculation. The value of \( n \) determines the class of the fluid:

\[ n = 1 \quad \text{Newtonian Fluid} \]

\[ n > 1 \quad \text{Shear thickening (dilatants fluid)} \]

\[ n < 1 \quad \text{Shear thinning (pseudo plastics)} \]

Input parameter values for non-Newtonian fluid are already mentioned in the Table 2.

In the present work, three dimensional orthogonal mesh was used with total 20000 hexahedral cells. Two dimensional representation of the grid is shown in the Fig 3.
The Mesh, model details and boundary conditions were directly taken from the ANSYS-12 user guide & manual [reference 8]. ANSYS-FLUENT 12 [reference 8] offers nine different turbulence models to capture the transition & turbulence effects in the flow. Since the present flow situation possibly lies in the transition flow situation thus standard K – \( \omega \) (Turbulence Kinetic Energy – Specific dissipation rate) model was adopted. This incorporated the modifications for shear flow corrections and low Reynolds number effects (within turbulence).

SIMPLE algorithm has been used for pressure velocity coupling. ANSYS – FLUENT12 [reference 8] uses a multi-grid scheme to accelerate the convergence of the solver by computing the corrections on a series of coarse grid levels. The use of multi-grid scheme can greatly reduce the number of iterations and the computational time required to obtain the converged solution.

A residual convergence of 10^{-6} has been obtained for the governing variables viz, mass balance, and velocity components of the flow, k and \( \omega \). Under relaxation parameters were kept moderate and constant throughout the solution as mentioned in Table 3 below.

<table>
<thead>
<tr>
<th>Parameters;</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>0.3</td>
</tr>
<tr>
<td>Momentum</td>
<td>0.7</td>
</tr>
<tr>
<td>Density</td>
<td>1</td>
</tr>
<tr>
<td>Body Forces</td>
<td>1</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>0.8</td>
</tr>
<tr>
<td>Turbulence Viscosity</td>
<td>1</td>
</tr>
<tr>
<td>Specific Dissipation Rate</td>
<td>0.8</td>
</tr>
<tr>
<td>Energy</td>
<td>1</td>
</tr>
</tbody>
</table>

For representation of the results the calculated values will be displayed on defined planes p1, p2 and p3. The three planes are at locations of 3 o’clock, 12 o’clock and 9 o’clock corresponding to analogy with hour-arm of watch with respect to geometry under study. These locations are depicted in the Figure 4 and Table 4 as below.

<table>
<thead>
<tr>
<th>Plane</th>
<th>Coordinate</th>
<th>Min Extent (m)</th>
<th>Max Extent (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>P1</td>
<td>X</td>
<td>0.01 m</td>
<td>0.015 m</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.00 m</td>
<td>0.00 m</td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>0.005 m</td>
<td>0.005 m</td>
</tr>
<tr>
<td>P2</td>
<td>X</td>
<td>-0.00515 m</td>
<td>-0.00515 m</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.00566585 m</td>
<td>0.02015 m</td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>0.005 m</td>
<td>0.005 m</td>
</tr>
<tr>
<td>P3</td>
<td>X</td>
<td>-0.0253 m</td>
<td>-0.01 m</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.00 m</td>
<td>0.00 m</td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>0.005 m</td>
<td>0.05 M</td>
</tr>
</tbody>
</table>

The results obtained were normalised only for the purpose of plot presentation of the results and that only of the velocity values. All velocity values were normalised with respect to inlet velocity as given below;

\[
U_{an} = \frac{U_a}{U_{inlet}}, U_{tn} = \frac{U_t}{U_{inlet}}, \text{ and } U_{rn} = \frac{U_r}{U_{inlet}}.
\]

Distances along reference planes p1, p2 and p3 for plotting were normalised with respect to the dimension of eccentricity along these respective planes. These are;

\[
p_{1nX} = \frac{p_{1X(\text{max})} - p_{1X(\text{min})}}{0.015} = \frac{0.015 - p_{1X}}{0.005}.
\]

\[
p_{2nY} = \frac{p_{2Y(\text{max})} - p_{2Y(\text{min})}}{0.02} = \frac{0.02 - p_{2Y}}{0.012}.
\]

\[
p_{3nX} = \frac{p_{3X(\text{max})} - p_{3X(\text{min})}}{0.05} = \frac{0.05 - p_{3X}}{-0.0253} = 0.0153.
\]

Here, subscripts ‘max’ and ‘min’ indicate the respective maximum and minimum values as mentioned in the Table 4.

### III. VALIDATION

The predicted flow field is validated against the experimental data by J. M. Nouri et al [1]. For this, axial velocity (\( U_a \)) and tangential velocity (\( U_t \)) values were chosen. These were plotted along the three specified planes p1, p2 and p3 with respect to geometry under study. These locations are depicted & detailed in the Figure 4 and Table 4. All velocity values and the specified planes have been normalised (already discussed) for the representation.

Figure 5 says that the comparison between experimental and numerical result (by QUICK scheme) for variation of the axial velocity in the annular gap along the plane p1. There is qualitative match found between the experimental and numerical solutions, and have same trend of variation of axial.
velocity. Figure 6 is representing the comparison between the Experimental and Numerical results (by QUICK Scheme) for variation of the tangential velocity along the plane p1. Results by the two methods are following the same trend of variation of tangential velocity values. Both results are satisfying the physical situation of the problem as well.

Figure 7 and Figure 9 are representing the axial velocity variation along planes p2 and p3 respectively for experimental and numerical results (by QUICK scheme). Figure 8 and Figure 10 are representing the tangential velocity variation along planes p2 and p3 respectively for experimental and numerical results (by QUICK scheme). By visual examination a qualitative match in the trend of velocity variation can be seen in these results also.

The computed and experimental results are observed to be in good agreement.

IV. RESULTS AND DISCUSSIONS

The detailed results of the flow are presented using different types of differencing schemes viz. second order upwind, Power law differencing scheme and QUICK differencing scheme. Axial velocity, Radial velocity and Tangential velocity values of fluid particles were plotted for these three different schemes and compared with experimental data in Figure 5 to 10. Also contours for molecular velocity variation and turbulence kinetic energy variation, at
middle z-plane (i.e. at $z = 0.005$ m) of the model for various schemes have been presented in Figure 11 – 19. These physical quantities are very crucial in transition flow study for Transition non-Newtonian fluid flow.

Molecular viscosity contours, figure 11 – 13, indicate that its value is maximum at around middle of the annular gap, at all values of the radial coordinate. Then, as wall is being approached may it be inner part of outer cylinder (which is stationary) or outer portion of the inner cylinder (which is in rotation at constant angular speed) molecular viscosity is approaching its minimum value. Molecular viscosity varies in the similar fashion as dynamics viscosity. Thus molecular viscosity is inversely proportional to rate of shear strain. Since for a transition flow situation, in the middle portions of the flow gaps shear strain rate is least in value thus naturally this results in maximum value of the molecular viscosity value.

Turbulence kinetic energy (TKE) is the mean kinetic energy per unit mass associated with eddies in turbulent flow. Physically, the turbulence kinetic energy is characterised by measured root-mean-square (RMS) velocity fluctuations. In Reynolds-averaged Navier Stokes equations, the turbulence kinetic energy can be calculated based on the closure method, i.e. a turbulence model. Generally, the TKE can be quantified by the mean of the turbulence normal stresses:

$$k = \frac{1}{2} \left\{ (\overline{u'_a})^2 + (\overline{u'_b})^2 + (\overline{u'_c})^2 \right\}$$

Normal stress values being very small (as the fluctuating components of velocities being very small) turbulence kinetic energy will be small or vice versa. As indicated by the turbulence kinetic energy contours, Figure 14 – 16, being maximum near walls and reduces to minimum while proceeding towards middle portions of the eccentric gap at whichever radial direction considered. This shows that at these portions normal stress values and so fluctuating components of velocities and thus shear stress values will be maximum or minimum at those respective portions of the flow field.

Now as per the above discussion, when comparing the molecular viscosity contours, Figure 11 – 13, and turbulence kinetic energy contours, Figure 14 – 16, the two results are very well consistent as per the theory of fluid dynamics.
Axial velocity contours, Figure 17 – 19, indicate maximum values of it at lower part of the maximum eccentric gap in the flow domain. Now, proceeding towards the walls, it approaches zero velocity value satisfying the no-slip criterion at the solid boundaries because none of the cylinders is in axial motion (even though inner cylinder is in rotation).

Contour plots, from Figure 11 to 19, indicate that there is no significant difference found amongst the plots obtained by the three chosen schemes i.e. Power law scheme, QUICK scheme and second order upwind scheme. Qualitatively i.e. in terms of variation trend on the chosen plane \( z = 0.005 \) all three schemes match. Of course in terms of the range (Maximum to minimum value) of the value there is very slight difference, whether the case is of molecular viscosity contours or of turbulence kinetic energy contours or that of axial velocity. In some cases, these results of any ‘two’ schemes (different ‘two’ in different cases) are matching.
(A) towards inner cylinder (B). After distance coordinate of 0.4 differences between the predicted radial velocity values reduces for the schemes. Although magnitude wise there is difference, but variation of radial velocity for different schemes are following similar pattern.

Figure (20): Axial Velocity variation along p1 for different schemes

Figure (21): Axial velocity variation along p2 for different schemes

Figure (22): Axial velocity variation along p3 for different schemes

Figure (23): Radial velocity variation along p1 for different schemes

Figure (24): Radial velocity variation along p2 for different schemes

Figure (25): Radial velocity variation along p3 for different schemes

Figure (26): Tangential velocity variation along p1 for different schemes

Figure (27): Tangential velocity variation along p2 for different schemes
Figures 23 – 25, which are representing radial velocity variation along p1 and p3 respectively are showing the possibility of radial flow reversals by observing the predictions of QUICK and 2nd order upwind schemes, where as that of power law this possibility looks very less. This nature is quite surprising, firstly in the sense that why there is no flow reversal possibility along p2, and secondly in the sense that although power law is considered to be of accuracy between 2nd order and QUICK still it lagged behind in capturing the convective effects of flow. These are the points to be focussed upon.

V. CONCLUSION

This paper started with the objective of predicting the effectiveness of second order upwind and power law differencing scheme in respect to QUICK scheme for non Newtonian fluid flow through an eccentric annulus. In this, for speed of rotation of 300 rpm under a constant Reynolds no 9000 flow of a test fluid (of non Newtonian in nature as found that of drilling mud) was considered.

Results are very encouraging because there was a close match amongst the results obtained by all the considered differencing schemes. Of course second order scheme is much closer to the QUICK scheme which is considered very accurate in terms of prediction of transition flow, which is the type of flow under consideration. Further an experimental verification may be required of this numerical prediction.

ACKNOWLEDGEMENT

Authors acknowledge the cooperation extended by authorities of National Institute of Technology, Raipur (C.G.) INDIA for providing the licensed commercial software code ANSYS FLUENT (ANSYS Academic Research CFD version 12). Also we acknowledge TecplotInc for providing Demo version of Tecplot360 for plotting some of the diagrams.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Notation</th>
<th>Description of variables/Constants</th>
</tr>
</thead>
<tbody>
<tr>
<td>D_a</td>
<td>Hydraulic mean Diameter, 26 m</td>
</tr>
<tr>
<td>E</td>
<td>Eccentricity (Displacement of inner –cylinder axis from outer-cylinder axis), m</td>
</tr>
<tr>
<td>e_max</td>
<td>Maximum eccentricity, m</td>
</tr>
<tr>
<td>K</td>
<td>Consistency Index, Pa-s²</td>
</tr>
<tr>
<td>n</td>
<td>Power-law index</td>
</tr>
<tr>
<td>P</td>
<td>Pressure, Pa</td>
</tr>
<tr>
<td>R</td>
<td>Radial distance from axis of inner-cylinder, m</td>
</tr>
<tr>
<td>R_i, D_o</td>
<td>Outer radius &amp; diameter of inner-cylinder, m</td>
</tr>
<tr>
<td>R_i, D_o</td>
<td>Inner radius &amp; diameter of outer cylinder, m</td>
</tr>
<tr>
<td>A</td>
<td>Cross sectional area of annulus of flow, (π/4) (D_o² – D_i²) m²</td>
</tr>
<tr>
<td>Re</td>
<td>Bulk axial Reynolds number, 26pU/µwall</td>
</tr>
<tr>
<td>U</td>
<td>Bulk axial velocity, m/s</td>
</tr>
<tr>
<td>U_inle</td>
<td>Bulk axial velocity at inlet, m/s</td>
</tr>
<tr>
<td>U_1</td>
<td>Axial component of the velocity, m/s</td>
</tr>
<tr>
<td>U_2</td>
<td>Tangential component of velocity, m/s</td>
</tr>
<tr>
<td>U_3</td>
<td>Radial component of velocity, m/s</td>
</tr>
<tr>
<td>U_n</td>
<td>Normalised radial velocity</td>
</tr>
<tr>
<td>U_m</td>
<td>Normalised tangential velocity</td>
</tr>
<tr>
<td>U_o</td>
<td>Normalised axial velocity</td>
</tr>
<tr>
<td>z</td>
<td>Axial distance, m</td>
</tr>
<tr>
<td>φ</td>
<td>Coordinate along axis (i.e. z-direction)</td>
</tr>
<tr>
<td>Ω</td>
<td>Angular location with respect to inner cylinder</td>
</tr>
<tr>
<td>T</td>
<td>Coordinate along angular direction (i.e. tangential coordinate)</td>
</tr>
<tr>
<td>ρ</td>
<td>Fluid Density, kg/m³</td>
</tr>
<tr>
<td>µ</td>
<td>Characteristic Dynamic viscosity for flow, Pa-s</td>
</tr>
<tr>
<td>µ_wall</td>
<td>µ at wall, Pa-s</td>
</tr>
<tr>
<td>D</td>
<td>Angular velocity of inner cylinder, rad/s</td>
</tr>
<tr>
<td>O_i</td>
<td>Centre of inner cylinder (reference centre)</td>
</tr>
<tr>
<td>O_o</td>
<td>Centre of the outer cylinder</td>
</tr>
<tr>
<td>T</td>
<td>Time coordinate</td>
</tr>
<tr>
<td>σ_z</td>
<td>Characteristic Shear Rate of flow (s⁻¹)</td>
</tr>
<tr>
<td>p</td>
<td>Flow pressure</td>
</tr>
<tr>
<td>e_r</td>
<td>Unit vector along radial coordinate</td>
</tr>
<tr>
<td>e_θ</td>
<td>Unit vector along tangential coordinate</td>
</tr>
<tr>
<td>e_z</td>
<td>Unit vector along axial coordinate</td>
</tr>
<tr>
<td>P1_x</td>
<td>Distance along plane P1</td>
</tr>
<tr>
<td>P2_x</td>
<td>Distance along plane P2</td>
</tr>
<tr>
<td>P3_x</td>
<td>Distance along plane P3</td>
</tr>
<tr>
<td>P1_x, y</td>
<td>Normalised values in relation to P1_x</td>
</tr>
<tr>
<td>P2_x, y</td>
<td>Normalised values in relation to P2_x</td>
</tr>
<tr>
<td>P3_x, y</td>
<td>Normalised values in relation to P3_x</td>
</tr>
</tbody>
</table>

REFERENCES


Comparison of Various Numerical Differencing Schemes in Predicting Non-Newtonian Transition flow through an Eccentric Annulus with Inner Cylinder in Rotation


[5]. V. C. Kelessidis (SPE) and G. E. Bandelis (Technical U. Of Crete), Flow Pattern and Minimum Suspension Velocity for Efficient Cuttings Transport in Horizontal and Deviated Wells in Coiled Tube Drilling, December 2004 SPE (Society of Petroleum Engineers), Drilling and Completion.


[15]. Sang-mok Han, Nam-sub Woo, and Young-kyu Hwang; Solid-liquid mixture flow through a slim hole annulus with rotating inner cylinder; Journal of Mechanical Science and Technology 23 (2009) 569–577.