

EXPERIMENTAL STUDIES ON PASSIVE COOLING SYSTEM OF HEAT GENERATING SOURCE

Submitted in partial fulfilment of the requirements for the award of
Bachelor of Technology Degree in Chemical Engineering

By

SHANOOP.P. (40690013)

SOM NATH YADAV (40690014)



**DEPARTMENT OF CHEMICAL ENGINEERING
SCHOOL OF BIO & CHEMICAL ENGINEERING**

SATHYABAMA

**INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)**

**Accredited with Grade “A++” by NAAC
JEPPIAAR NAGAR, RAJIV GANDHI SALAI, CHENNAI-600119**

OCTOBER-2023



SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited "A++" Grade by NAAC | 12B Status by UGC | Approved by AICTE

www.sathyabama.ac.in

DEPARTMENT OF CHEMICAL ENGINEERING

BONAFIDE CERTIFICATE

This is certify that this Project Report is the bonafide work of **SHANOOP.P. (Reg No 40690013)** and **SOM NATH YADAV (Reg No 40690014)** who carried out the project entitled "**EXPERIMENTAL STUDIES ON PASSIVE COOLING SYSTEM OF HEAT GENERATING SOURCE** " under our supervision from July 2023 to October 2023.

Chenman
Internal Guide

Dr. D. VENKATESAN, M.Tech., Ph.D.

Associate Professor

[Signature]
External Guide

Dr. G. SUNEEL, M.Tech, Ph.D.

Supdt.(O) WIP,UOF & VWSF

NRB.BARCF Kalpakkam

[Signature]
Head of the Department

Dr. S. SATHISH, M.E, Ph.D

Associate Professor

HEAD OF THE DEPARTMENT
DEPARTMENT OF CHEMICAL ENGINEERING
SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Jeppiaar Nagar, Rajiv Gandhi Salai
Chennai - 600 115

Submitted Viva voce Examination held on

4-11-23

N. 880884
Internal Examiner

[Signature]
External Examiner



Bhabha Atomic Research Centre
Nuclear Recycle Board
Integrated Nuclear Recycle Plant (K)
Waste Immobilisation Plant

Kalpakkam 603102
Intercom: 43761
Email: suneel@ignrc.gov.in

Ref: WIP/OP.01/2023/

Date :12.06.2023

To,

Dr. D. Satish,

HOD, Department of Chemical Engineering,

Satyabhama University, Rajiv Gandhi Salai,

Chennai

Sub: Project acceptance approval reg:

This is to certify that this Project Report entitled “**Experimental Studies on passive cooling system of heat generating source**” is a bonafide work of **Mr. SHANOOP.P (Reg No. 40690013)** in partial fulfilment of the requirements for the award of “**Bachelor of Technology in Chemical Engineering**” who can carry out the project under my supervision from June 2023 to November 2023. After the completion of the project, a journal publication of the work carried out shall be done.

With regards,

G. Suneel

Supdt. (O), WIP, UOF and VWSF

जी. सुनील/ G.SUNEEL
अधीक्षक (ओ)/ Superintendent (O)
(इन्सुओपी, वीटव्हाइसएफ एंड यूओएफ) (WIP, VWSF & UOF)
खरिएनसारीके, एनआरसी, गीरसारीएफ/ INRPK, NRB, BARC
कल्पाक्कम/ Kalpakkam – 603102



Bhabha Atomic Research Centre
Nuclear Recycle Board
Integrated Nuclear Recycle Plant (K)
Waste Immobilisation Plant

Kalpakkam - 603102
Intarcom - 43761
Email: suneel@ignrp.gov.in

Ref: WIP/OP.01/2023/

Date : 12.06.2023

To,

Dr. D. Satish,

HOD, Department of Chemical Engineering,

Satyabhama University, Rajiv Gandhi Salai,

Chennai

Sub: Project acceptance approval reg:

This is to certify that this Project Report entitled “**Experimental Studies on passive cooling system of heat generating source**” is a bonafide work of **Mr. Somnath Yadav (Reg No. 40690014)** in partial fulfillment of the requirements for the award of “**Bachelor of Technology in Chemical Engineering**” who can carry out the project under my supervision from June 2023 to November 2023. After the completion of the project, a journal publication of the work carried out shall be done.

With regards,

G. Suneel

Supdt. (O), WIP, UOF and VWSF

जी. सुनील / G.SUNEEL

अधीक्षक (ओ) / Superintendent (O)
[सुपरिन्टेन्डेंट, बीएनएआरएचसीए/INRP, VWSF & UOF]
बीएनएआरएचसीए, राजीव गांधी, INRP, NRB, BARC
कल्पक्कम/ Kalpakkam – 603102

DECLARATION

I SHANOOP.P. (Reg. No. 40690013) hereby declare that the Project Report entitled "EXPERIMENTAL STUDIES ON PASSIVE COOLOING SYSTEM OF HEAT GENERATING SOURCE" done by me under the guidance of Dr. D. VENKATEAN, M.Tech., Ph.D., Associate Professor, Department of Chemical Engineering, Sathyabama, Institute of Science and Technology and Dr. G. SUNEEL, M.Tech., Ph.D., Supdt. (O) WIP, UOF& VWSF Bhabha Atomic Research Centre kalpakkam is submitted in partial fulfilment of the requirements for the award of Bachelor of Technology in Chemical Engineering.

DATE: 31-10-2023

PLACE: Chennai

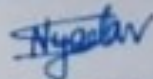
SIGNATURE OF THE CANDIDATE

DECLARATION

I **SOM NATH YADAV**, (Reg. No. 40690014) hereby declare that the Project Report entitled "**EXPERIMENTAL STUDIES ON PASSIVE COOLING SYSTEM OF HEAT GENERATING SOURCE**" done by me under the guidance of **Dr. D. VENKATESAN, M.Tech., Ph.D.**, Associate Professor, Department of Chemical Engineering, Sathyabama, Institute of Science and Technology and **Dr. G. SUNEEL, M.Tech., Ph.D.**, Supdt. (O) WIP, UOF& VWSF Bhabha Atomic Research Centre Kalpakkam is submitted in partial fulfilment of the requirements for the award of Bachelor of Technology in **Chemical Engineering**.

DATE: 31-10-2023

PLACE: Chennai



SIGNATURE OF THE CANDIDATE

ACKNOWLEDGEMENT

We are pleased to acknowledge our sincere thanks to **The Board of Management** of SATHYABAMA for their kind encouragement in doing this project and for completing it successfully. We are grateful to them.

We convey our thanks to **Dr. S. SATHISH, M.E., Ph.D.**, Head of the Department of Chemical Engineering for providing us necessary support and details at the right time during the progressive reviews.

We would like to express our sincere and deep sense of gratitude to our Project Guides **Dr. D. VENKATESAN, M.Tech, Ph.D.**, Associate Professor, Department of Chemical Engineering Sathyabama, Institute of Science and Technology and **Dr. G. SUNEEL, M.Tech., Ph.D.**, Supdt. (O) WIP, UOF & VWSF, Bhabha Atomic Research Centre Kalpakkam for his valuable guidance, suggestions and constant encouragement paved way for the successful completion of our project work.

We wish to express our thanks to all Teaching and Non-teaching staff members of the **Department of Chemical Engineering** who were helpful in many ways for the completion of the project.

SHANOOP P.

SOM NATH YADAV

ABSTRACT

Passive cooling techniques play a crucial role in the safe management of heat generated from radioactive canisters in nuclear waste storage facilities. The concept of passive cooling and its application for efficiently dissipating heat from these canisters without relying on active mechanical systems. The focus is on the utilization of natural heat transfer mechanisms and design strategies to enhance cooling effectiveness while minimizing energy consumption and maintenance requirements. It explores various design considerations such as heat sink materials, surface area optimization, and thermal insulation to achieve effective cooling while maintaining safety and regulatory requirements. It presents thermal analysis, temperature distribution profiles, and heat transfer simulations to validate the performance of passive cooling systems.

Passive cooling offers a promising approach for managing heat generated from radioactive canisters in nuclear waste storage. By harnessing natural heat transfer mechanisms, passive cooling systems provide a reliable and energy-efficient solution while ensuring the long-term safety and integrity of the storage facility. Continued research and development in this area will contribute to the advancement of passive cooling techniques and their widespread application in nuclear waste management practices.

TABLE OF CONTENTS

CHAPTER NO.	TITLE	PAGE NO
	ABSTRACT	I
	LIST OF ABBREVIATIONS	III
	LIST OF FIGURES	IV
	LIST OF TABLES.....	V
1	INTRODUCTION	1
	1.1. Radioactive Waste Management	1
	1.2. Strategy for HLW management.....	2
	1.2.1. Immobilisation of HLW.....	2
	1.2.2. Interimstorage	3
	1.2.3. Final Disposal	4
2	LITERATURE SURVEY	7
	2.1. Passive cooling system of heat generating source:	7
3	AIM AND SCOPE	18
4	EXPERIMENTAL DESCRIPTION	19
	4.1. Experimental setup for Calorimetric Studies	19
	4.2. Procedure for calorimetry Analysis of VWP Canister	21
	4.3. Wind velocity measurement	24
	4.4. Development of Two Dimensional Model	24
5	RESULTS AND DISCUSSION	32
6	SUMMARY AND CONCLUSIONS	38
	REFERENCES	39

LIST OF ABBREVIATIONS

HLW	High-level Liquid Waste
VWP	Vitrified Waste Product
VWSF	Vitrified Waste Storage Facility
HEPA	High-Efficiency Particulate Absorbing filter

LIST OF SYMBOLS

ρ	Density
C_p	Specific Heat
β	Coefficient of volumetric expansion
g	Acceleration due to gravity
k_a	Conductivity of air
μ	Coefficient of Viscosity of air
k_g	Conductivity of Glass
h	heat transfer coefficient
A	Surface Area

LIST OF FIGURES

FIG. NO.	TITLE	PAGE NO.
1.1	Joule Heated Ceramic Melter	3
1.2	interim storage	5
1.3	Deep geological repository	6
2.1	Vitrified Waste Storage Facility	8
4.1	Calorimetry setup	19
4.2	Schematic of the Calorimetry at VWSF	20
4.3	Canister loading	22
4.4	Vault area	22
4.5	Model algorithm	31
5.1	Results of calorimetry	32
5.2	Relation between mass flow rate and wind speed	33
5.3	Glass temperature profile without cooling	33
5.4	Temperature profile for 2 kW decay heat with passive cooling	34
5.5	Velocity profile across ventilation pipe for 2 kW decay heat	34
5.6	Temperature profile for 4 kW decay heat with passive cooling	35
5.7	Velocity profile across ventilation pipe for 4 kW decay heat	35
5.8	Temperature profile for 6 kW decay heat with passive cooling	36
5.9	Velocity profile across ventilation pipe for 6 kW decay heat	36
5.10	Temperature profile for 8 kW decay heat with passive cooling	37
5.11	Velocity profile across ventilation pipe for 8 kW decay heat	37

LIST OF TABLES

Table No.	Title	Page No.
4.1	Calorimetric observation	22
4.2	Parameters at different heat loading conditions	23
4.3	Flow Measurement	23
4.4	Parameters used for calculating wind speed	24
4.5	VWSF Storage details	30
4.6	Dimensions of Flow Region	30
5.1	Experimental results of the calorimetry	32

1 INTRODUCTION

1.1. Radioactive Waste Management

Coal, natural gas, and oil are the three main source of energy used worldwide. Since these resources are not replenishable, they are unable to meet the rising energy demand. In addition, the worry over the greenhouse gas emissions caused by the continued use of fossil fuels necessitates the use of alternative energy sources to lessen the negative effects indicated above. Due to its low carbon footprint, nuclear power can serve as a catalyst for the shift to sustainable energy.

The production of radioactive waste is a side effect of nuclear energy utilisation. These radioactive wastes generate heat, necessitating regular cooling and observation. Immobilisation is the first step in the HLW management approach, which is followed by intermediate storage and disposal. By vitrifying HLW into glass matrix with a Joule-heated ceramic melter, immobilisation is accomplished. Glass and HLW are melted together, and the mixture is then placed into SS304L canisters. Up to 2 kW of decay heat can be produced by a single canister. To lessen the heat produced by radioactive decay, the Vitrified Waste Product (VWP) must be monitored for 30 years.

Vitrified Waste Storage Facility (VWSF) facilitates the storage of VWP. The centre line temperature of VWP should always be maintained below the transition temperature of borosilicate glass that is $\sim 500^{\circ}\text{C}$. The cooling of VWP for a prolonged time is achieved by a natural draft cooling system consisting of a 100 m high stack, which ensures that air flows axially between the ventilation pipe and the thimble tube. Cold air enters the vault via the inlet air corridor and is distributed in a ventilation pipe via the supply plenum at the bottom of the vault. Four canisters can be stored inside the thimble tube. The thimble tube is 11 m long and hung from the top wall of the vault. All storage locations have a lower plenum where air is fed from the inlet duct. This air flows axially via the annular space between the thimble and sleeve. At the top of plenum, the flow that emerges from the top of each storage location is merged before going to the stack. Due to buoyancy, the air flow in the annular space between the ventilation pipe and thimble warms up and rises. Due to the chimney effect, the hot air rises through the stack and keeps the flow through the vault going. The sleeve serves as both a stack and an air flow guide.

1.2.Strategy for HLW management

High level radioactive liquid waste (HLW), which includes the vast majority (99%) of the radioactivity in the whole fuel cycle, is produced during the reprocessing of spent fuel. In addition, after the spent fuel is dissolved for reprocessing, hull waste, or the hollow clad tubes, is formed as solid HLW. The safe treatment of radioactive waste, particularly the HLW, is a key component of public support for nuclear energy. In order to effectively manage HLW, it must be isolated from the biosphere and subjected to long-term monitoring that spans multiple generations. Therefore, in the context of India, the management of HLW includes the following three steps.

1.2.1.Immobilisation of HLW

One of the few nations with expertise in vitrification technology is India. In specially constructed "hot cells "made up of 1.5-meter-thick concrete wall due to the high radiation fields, various operations are remotely carried out. These hot cells have systems and tools for remote handling. Custom-made robots, manipulators, remote welding units and remote surveillance tools are a few of the main remonetisation gadgets. Indian industries, national organizations and academic institutions have actively collaborated on the indigenous development of the remote handling equipment. Glass matrix development for HLW depends on a combination of processing temperatures, specific glass additives, and composition. The achievement maximum loading of waste into glass is constrained by the heat of decay and the solubility of the trash's constituents. Glass-forming additives must adhere to permitted processing temperatures and chemical resistance. These processing temperatures are determined by the radionuclide's particular volatility and the melter material's compatibility with the corrosive environment of molten glass. Due to their restricted solubility or incompatibility with glass composition, the presence of specific chemical species in high level waste, such as thorium, aluminium, fluorine, sulphate, platinum group metals, etc., presents extra challenges for the creation of glass formulations. Utilising cutting-edge analytical tools, the vitrified products are assessed for a number of parameters including chemical durability, radiation stability, melt temperature, homogeneity, waste loading, and thermal stability. The solidified waste form must also satisfy the requirements for short-term and long-term storage before being finally disposed of in a deep geological repository. Figure 1.1

depicts the layout of the ceramic melter heated by joules principle. Indian programme for managing radioactive waste is vitrifying HLW by using melters. The VWP is sealed in appropriate containers and over packs and stored for around 30 years under observation and to allow radioactive decay heat to dissipate. Enough information about the behaviour of the product would be produced over this period of observation. Additionally, it is anticipated that the product's radiation and heat conditions will stabilise to the point where transporting it is practical. A natural flow air conditioning has been created for the vault based on safety and intricate techno-economic factors.

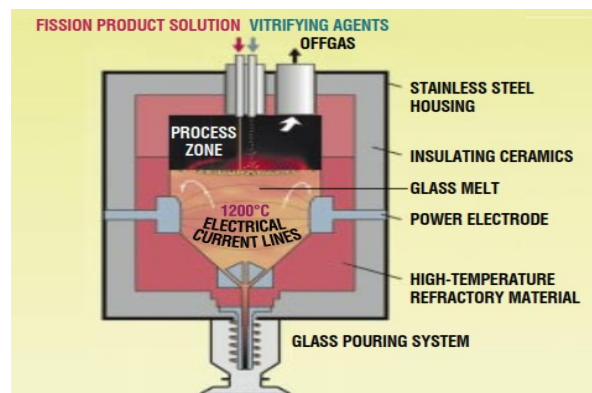


Figure 1.1: Joule Heated Ceramic Melter

1.2.2. Interim storage

For temporary storage, a number of various designs have been taken into account. The use of water ponds as temporary storage for the solidified wastes is merely an expansion of the current method of spent fuel storage. Demineralization and ongoing chemical balancing of the water are required. Both air cooling and water cooling are deemed to be acceptable in terms of safety and thorough technological and economic analysis, however water cooling has a few extra benefits. A superior coolant, water also offers complete activity containment because of its high heat capacity and thermal conductivity compared to air, the canister may fail. However, some drawbacks include corrosion problems and a high make up water need. In India, air-cooled storage is used as a result of a thorough techno-economic analysis.

In India, the passive air-cooling design by natural convection is supplemented by induced draught from the stack, which is independent of people or machinery, is used to store solidified high-level waste. It is using the decay heat and a well-

constructed stack to propel air through the storage vault. In practice, the waste heat will raise the air's temperature, which will cause the air to rise as a result of buoyant forces. The driving force brought on by buoyancy is counterbalanced by the friction effect of air passing through the system in order to reach equilibrium for the loading state of each compartment of the vault.

Certain fundamental requirements that, in certain situations, are in conflict must be balanced in the structural facility design. The project site has a higher ground water table, which is projected to surge up to virtually surface level in the monsoon season. Optimum wall thickness is required for the vault to ensure water tightness and constant structural stability against uplift. To minimize heat effects, however, standard concrete must be used for the structure. Only by minimizing the temperature gradients along the concrete wall, which is possible by decreasing the thickness of the vault wall, can this be accomplished. To meet the aforementioned standards, it is essential to isolate the thermal effects from the load-bearing structure as well as any water infiltration. A double vault design is used to accomplish this. The design of the inner storage vault is based on thermal factors, whereas the design of the outer vault is based on biological shielding structural concerns. Additionally, the outer vault would keep groundwater seepage from the local area around the storage structure. Water from seepage should be further processed and activity will be watched for. From the outer vault, specially made bearings support the thermal vault. This enables the thermal vault to slide freely, alleviating the thermal vault of temperature strains.

1.2.3. Final Disposal

International agreement has evolved that deep geological disposal is the best suitable method for permanently separating of vitrified high-level waste from the environment. Remoteness from the surrounding area, the lack of ground water circulation and the capacity to store radio nuclides for geologically long times are the three fundamental requirements for a geological formation to be acceptable for the installation of a facility for disposing of radioactive waste. India offers a diverse range of rock types, especially those with a strong capacity to act as a natural barrier to confine and separate VWP. The most likely possibility for a deep geological deposit could be granites, which make up 20% of the country's overall surface.

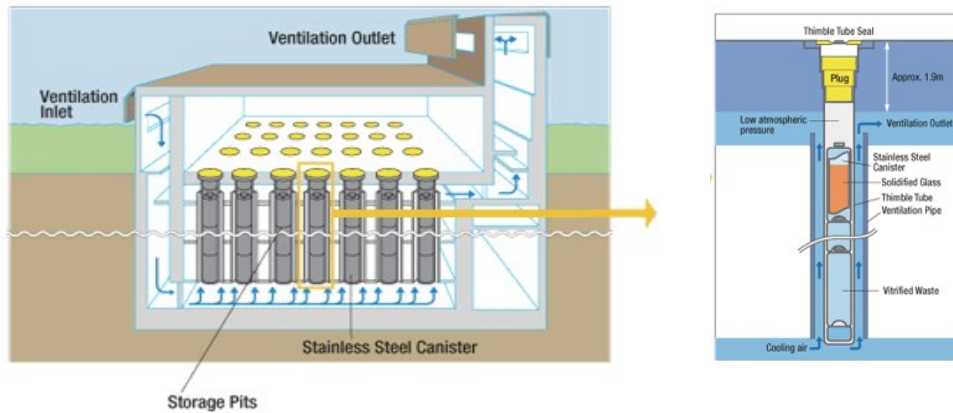


Figure 1.2: Interim storage

Even though India won't need a deep geological repository for several decades, work is already being done in the areas of characterization of numerical modelling, natural barriers, conceptual design, and natural analogs of waste forms and repository procedures. To provide more separation, radioactive waste is disposed of using a system of several barriers. The total safety against radioactive migration is achieved through the careful selection of waste form, an appropriate engineered barrier, backfill, and site-specific geo environmental factors. The most crucial elements of the multi barrier strategy used in a deep geological disposal system in hard rocks are backfills and buffers. These are positioned as layers between the host rock and the waste over pack primarily to prevent groundwater from flowing toward the waste form and to slow radionuclide migration into the biosphere in the improbable event that the waste over pack were to be breached and release radio nuclides. Bentonitic clays that swell have become more popular options for backfill. For the purpose of evaluating the safety of disposal facilities under different scenarios, model design, execution, and data are crucial. Such safety evaluations are founded on a thorough physical comprehension of the processes impacting the repository and the geological formation, as well as those engaged in the transportation and release of radio nuclides. Diagram of the multi-barrier disposal idea, Figure 1.3. Indian program for managing radioactive waste is reuse and recycle technology development and the desire to efficiently use resources have given rise to possibilities of recycling options that could lighten the load on future generations. Through enhanced plutonium recovery and recycling will considerably decrease the radioactivity of the waste. HLW's long-term radio-toxicity to be less of an environmental burden for nuclear power to continue to

grow. The long-lived minor actinides (Am, Np, Cm) and fission products (I-129, Tc-99, etc.) are separated from the radioactive waste in the partitioning and transmutation process, and then they are either rendered non-radioactive or transformed into elements that had a lot shorter half-lives than the original elements. In order to eliminate or decrease radioactive stockpiles, this process may be accomplished in Accelerator Driven Sub-Critical Systems (ADSS) or Integral Fast Reactors (IFR). This long-term approach to managing high-level trash would benefit the environment and the availability of resources.

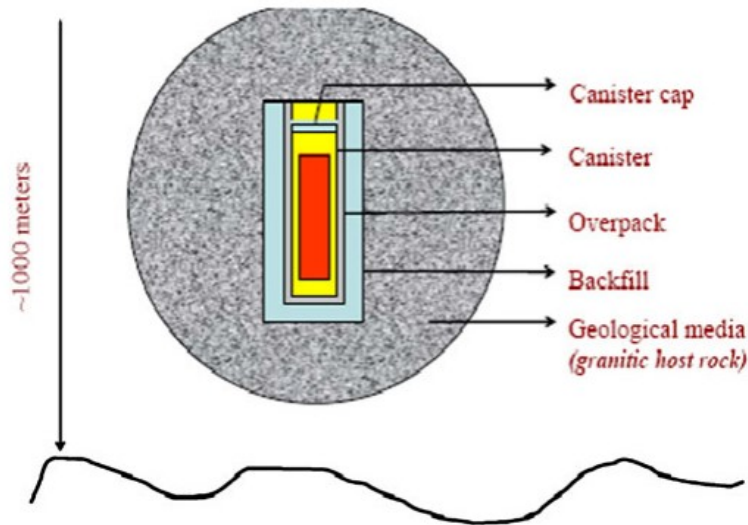


Figure 1.3: Deep geological repository

Partitioning of HLW also enables the conditioning of specific radioactive waste streams with the available vitrification technologies using cutting-edge ceramic waste forms like Synroc. Because it is poly phase and polycrystalline, Synroc has an extra benefit for immobilizing high actinide waste loadings. India is working to attain the aforementioned goals through a development program.

2 LITERATURE SURVEY

2.1. Passive cooling system of heat generating source:

The Vitrified Waste Storage Facility (VWSF) is a facility created to store vitrified high-level waste in retrievable engineered storage canisters. The building is designed to effectively cool product canisters, keeping their temperatures within safe and acceptable limits. VWSF has a 2100 canister storage capacity. Each of the four storage vaults in the VWSF Building can accommodate both common and uncommon canisters.

Using the stack effect, the vault's cooling air is released normally during ventilation without being filtered. Induced ventilation with a High-Efficiency Particulate Absorbing filter (HEPA) is offered if airborne activity is discovered in the exhaust air. Before permanently disposing of the canister in a repository, it is imperative to ensure the integrity of the canister and the product. The heat generated by the disintegration of radioactive nuclides must be dissipated in order to prevent the waste product from melting down under any foreseeable circumstances. Following that, the canister would be qualified for final disposal. The decomposition heat from the canisters would be removed by continuous air movement brought on by natural convection and aided by a stack.

The thimble tubes that house the canisters are held up by a vault roof. There are four canisters piled inside of a thimble tube. Axial airflow is made possible by the air passing around the thimble in a ventilation pipe. Air inlet plenum connects ventilation pipes. An air supply intake tunnel leads air into the vault. It uses an exhaust plenum to leave the vault and enter the stack. Air is channelled axially through a ventilation pipe located around the canisters to provide cooling. If airborne activity is found in the exhaust before it reaches the stack, separate RCC plenums, fans, and filter banks are offered for air filtering. Under normal operating circumstances, there shouldn't be any activity in the exhaust air because air doesn't come into direct touch with the glass product. Table 4.2 displays the precise geometry of ventilation pipes and thimble tubes.

Mishra et al., 2022 carried out a numerical study of the radioactive fuel storage vault with the goal of enhancing ventilation performance by strategically placing interconnected ducts. To decide the layout of numerous connecting ducts, they examined the air flow through the vault. The heat removal efficiency and air

exchange rate through the vault decreases as the aspect ratio of connecting ducts increased.

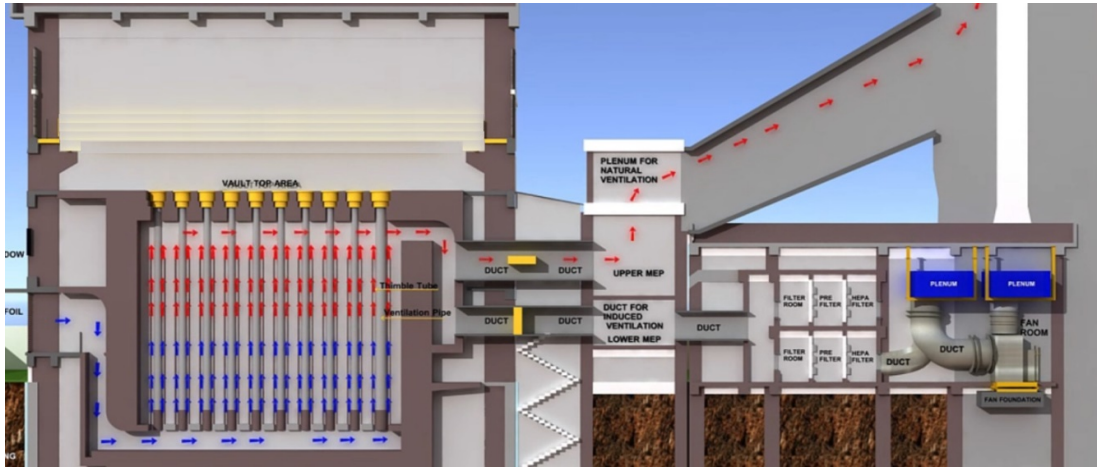


Figure 2.1: Vitrified Waste Storage Facility

Although circular vents give a higher air exchange rate, square ducts are good to eliminating heat. Through the reduction of flow losses (such as frictional losses and losses resulting from duct bends), the smoothing of duct edges enhanced air velocity. By attaching a nozzle to the supply ducts, the performance of the air jet penetration into the fuel pipe was improved. The addition of numerous little air supply nozzles was also shown to be more effective in removing heat than using a big duct. The jet above the heat load helped to better mix the air inside the vault.

CFD modelling was done for solid waste storage facilities by Verma in 2002. For the thermal research, a scaled-down version of such a facility was taken into consideration. Comparisons between theory and experiment have been made. 1-D analysis appears sufficient for the given setup and condition of uniform heating in all the over packs. The VWP storage vault underwent a numerical examination by Verma et al. in 2013. The parameters, such as duct and stack diameters, plenum height, etc., were studied parametrically. To get a thorough canister temperature profile, CFD code was employed. Outcomes of natural Analysis was done on the convection inside the canister and between the canister and thimble. It was found that the temperature reduces by 20 0C at the canister centre line.

M. Wang & Chen conducted studies to investigate the effectiveness of several turbulence models in an enclosed setting in 2009. The survey found that the inflow jet significantly increased space turbulence. The room's air turbulence was

lessened by the partition. The study found that while some RANS models did, others did not produce closer estimates for straightforward flows. The LES was the most accurate and trustworthy model. The DES model under-predicted the turbulent kinetic energy near the wall. If the sub-grid-scale turbulence kinetic energy is taken into account in the DES model, the result can be improved. The complex capabilities of the LES and DES models for handling airflow in enclosed environments are demonstrated in this work.

Chen et al., 2012 examined the RNG k- and SST k-'s propensity to forecast. An extensive experimental examination was carried out, predictions were produced using these RANS models, and the results were used to validate the numerical simulations. Both of the assessed turbulence models can satisfactorily capture the basic qualitative flow features, according to comparisons between experimental data and forecast results. The RNG k- model accurately predicts the maximum velocity decline as the jet reaches the floor, whereas the SST k- model performs well in the region of the impingement zone.

Numerical study was done on a square cavity with a heated floor and two slots—one for the entrance and the other for the outlet—by Koufi et al. in 2017. The locations of the outlet and inlet were adjusted and analysed in order to assess the ventilation effectiveness for temperature dispersion, define the optimal design offering, and determine the required thermal comfort. The governing equations were solved using the finite volume method, and the turbulence closure was performed using the RNG k-turbulence model. The SIMPLEC algorithm was applied to make sure the pressure-velocity correction was made. The layout that maintains thermal comfort the best is one in which the outflow is near the heated floor and the inflow is at the top.

Full-scale experiments and CFD analysis were used by Kuznik et al. in 2007 and he was examined in a mechanically ventilated enclosure, the fields of velocity and temperature. A close-to-the-ceiling axisymmetric jet is created by the ventilation system. The k- SST model, k- model, k- RNG model, and k- realizable model were contrasted in this study. None of the models could accurately predict the cold case. The global temperature and velocity fields, however, only displayed more accurate values in the hot and isothermal instances. Thus, in hot and isothermal conditions, the model can be used to forecast overall comfort as well as

the efficiency of thermal ventilation. Even though it does not exactly match the experimental data, the k- model was the most reliable for the cold environment.

CFD models of a dry storage cask were performed by Yoo et al. in 2010, and the outcomes were differentiated with the experimental data. The comparisons have a fair amount of accuracy. The peak guide tube's temperature variation is less than 1 K, which is within the experiment's thermocouples' limit. The size of the basket gap was found to be the most sensitive variable by the analysis.

Kim et al.'s 2014 investigation into heat transport in a concrete cask utilized for PWR spent fuel intermediate storage facilities. To safeguard the fuel cladding, which acts as a barrier for radioactive elements, the decay heat must be appropriately eliminated. To evaluate the design analysis process of the CFD tool, the experimental parameters were established by using half-scale model. The process for building of concrete cask half-scale model for the used dry fuel storage was established using scaling method. The CFD calculations successfully demonstrated a similar temperature rise between the exit and the inlet as well as the efficient maintenance of the exit temperature between the half-scale model and the prototype. The scaling ratios for its exit velocity and the air flow rate were similar with the CFD forecast.

Thermal investigation of accident circumstances in dry spent nuclear fuel storage facilities was done by Alyokhina in 2018. Numerical analysis was done on the flow structure inside ventilated containers. It was discovered what the main mass and heat transport mechanisms were for cooling spent nuclear fuel. Investigations were made into the thermal impacts of accident scenarios including circulated storage containers and channel obstruction. Heat removal in accident conditions is essential but more difficult than simply delivering air through input vents. In order to remove heat, outlet vents in a container's cooling system must be opened. The upper channel has to be opened first and ensure heat dissipation in the case of a complete ventilation duct blockage. For a typical occupied office room, Dominguez Espinosa & Glicksman, 2017 ran CFD simulations in spaces with high supply momentum to examine the impact of ventilation settings and inlet geometry on the air temperature profile in the room. The ventilation factors taken into account in this experiment were heat gain intensity and supply momentum. The height and area of the supply were changed to examine its reliance. The

Archimedes number, which contrasts the effects of buoyancy and supply momentum with dimensionless geometric variables, was used to characterize the configuration of the rooms. It was demonstrated that a high Archimedes space might be separated into two region: one where the temperature increases above the occupants in a warm region with uniform temperature in the other. The buoyancy effects on the flow of the space take precedence over the inertia of the air when the Archimedes number is high. The resulting thermal stratification profile is similar to ventilation displacement. The thermal plums convective effects mix the air as they move upward, creating a layer of warm air that is almost constant in temperature. In a typical Archimedes space, the supply jet mixes the air in the bottom section of the space, particularly near the outlet, making a result of stable temperature in this region. Because the supply jet was poor at mixing the air towards the ceiling, the temperature there was higher than it would have been with a higher Archimedes number. When the Archimedes number is low, the buoyancy effects o the heat sources are defeated by the momentum of the air entering space. Due to the high velocity mixing of the air there, which is extremely close to the outflow, the lower part of the room has an area of roughly equal temperature. For a given Archimedes number, the air temperature dropped in the bottom portion of the room while rising near the ceiling. The vertical mixing in the area benefited from a higher inlet. A bigger intake area results in temperature gradient in the room, with a lower part of the space being cooler and the floor area being warmer. The contour of the temperature profile was not appreciably altered by variable intake regions for samples with low and high Archimedes values. As predicted, it was demonstrated for high Archimedes numbers, the height of the entrance had no impact on the dimensionless temperature. For low Archimedes numbers, however, it had a more significant effect. When the Archimedes number is low, a high inlet lengthens the jet throw, extending the area where the supply air may mix. As a result, the temperature profile appeared to be more consistent, with the exception of regions near the ceiling and floor.

The performance of a dilution ventilation system was examined using CFD by Jiao et al., 2019. Effective dilution ventilation design is crucial for industrial hygiene control. A correctly constructed dilution ventilation system can increase the safety of tight workplaces while preserving comfortable working atmosphere. Due to the fact that the high- air flow area is only concentrated around the cross-

sectional area of air intake and outflow, single- inlet ventilation systems only partially reduce pollutant concentrations. The ventilation system's ability to reduce the amount of pollutants in small workplaces is thus diminished. The performance of the ventilation system is greatly enhanced by two-inlet installation configurations. It permits air to circulate broadly across the small workstation and cover a lot more surface than the standard method, which only has one intake and one outlet. The issue of uni-directional airflow within the small workspace can be successfully avoided by the opposite configuration of the air inlets, which can greatly reduce the pollutant concentration in the downwind area. The buoyancy-driven flows in industrial buildings that are largely ventilated by mechanical fans were examined by Mei et al. in 2018. In order to investigate naturally ventilated flows, an on-site measurement and a CFD model have been employed. The overall cooling needs can be met by the natural ventilation system, but local hotspots are formed at a number of sites because of the vortex airflow structure caused by buoyancy driven flow. To cool just a portion of the building, a cooling equipment using which is located in the same building that it provides cooling with supporting mechanical fans. To determine the ventilation flows, 3D steady-state Reynolds Averaged Navier-Stokes (RANS) equations are employed. On-site measurements were compared to the numerical codes and procedures. Using the average age of the air, natural ventilation's efficacy is evaluated. The average age of air is a mathematical measure of how long it takes a given quantity of air to travel through the system and reach a specific point. In this buoyancy-driven natural ventilation hall, an investigation is conducted into the possibility of employing slabs and mechanical vents to cool the equipment work area. The result of the simulation demonstrate that mechanical vents can successfully reduce air temperature and improve air exchange efficiently in the work area, considering that the ejection direction and velocity are highly related to the well- organized local airflow pattern.

A numerical analysis on air ventilation in the workshop room with different heat sources was carried out by Wiriyasart & Naphon, 2019. A 3-dimensional single-phase turbulent flow model was used in this study to conduct investigations to estimate thermal distribution and air quality in the workplace with diverse heat sources. The experimental findings are using to derive the beginning and boundary conditions. The air temperature and velocity in the workshop room were

found to be consistent with the experiment's results. The equipment placed in the workshop and the placement of the ventilation systems have a big impact on the suction force, which transfers air pollution and builds up heat inside the workplace.

A reduced-scale model investigation on the airflow distribution and cable heat dissipation in power cabins was carried out by J. Wang et al. in 2019. On the basis of the Archimedes number, a model at a scale reduction of 1:5 was constructed. The ventilation flow was solved using CFD simulations on the prototype power cabin using the three dimensional steady-state RANS equation and a realizable k- turbulence model. Due to airflow dispersion and cable architecture in the power cabin, the temperature distribution between the cables varies. The amount of turbulence at the air flow's entrance causes a significant drop in temperature.

A fabric drier was created, built, and put through an experimental test by Abdullah et al. in 2006. The results show that forced ventilation adding solar dryers have a higher drying rate than those with natural ventilation. The temperature increases in the solar cum forced-ventilation system is related to the internal heat flux induced by moisture removal and solar radiation, according to early computational calculations based on the Navier-Stokes thermal transient simulation technique. The corresponding experimental data and the numerical simulation are in good agreement. The feasibility of deploying more enormous fans was investigated using parametric research. In especially during the initial drying phases, higher ventilation rates may increase the dryer's performance through an improve temperature rise and in local heat flow.

An experiment was done to investigate the thermal performances of stratum ventilation by Tian et al. in 2011. To combat excessive indoor temperatures, strata ventilation has been advocated. In a strata-ventilated office building, tests on air velocity, temperature, and CO₂ content were done. Four different scenarios—two air volumes of 5.4 or 4.7 air changes per hour and two supply air temperatures (21 or 19⁰C) are tested in the experiments. The test room's HVAC system contains a damper that may be changed to control the rate of airflow. An orifice meter used to measure volumetric flow rate. Two no's of thermocouples are used to measure the supply air's temperature. In contrast to the two examples with inlet air temperature of 19 ⁰C, it is found that the two cases with inlet air temperatures of 21 ⁰C function better. In every case, the ventilation efficacy is around 1.5. In order to show

inclination and buoyancy impacts on transient mixed convection phenomena in a path with two equal open cubic cavities imposed with heat per unit area, Cárdenas et al., 2019 undertook an experiment. The walls of the holes towards the apertures are exposed to a continual wall heat flux, unlike the other boundary walls of the channel and the cavities, which are adiabatic and insulated. The experiments are run for a constant Pr 7, a range of buoyancy strengths or modified Richardson numbers from 32 to 300, a channel slope of 0 to 90 degrees, and a Reynolds number of 450 to 1450. The distributions of the nu and mean surface temperatures were averaged. The results demonstrate that, based on the duct direction and buoyancy intensity, the temperature distributions and flow exhibit an undulating pattern and were not equal for the duct centre-line. At high Ri values, the heat transfer rates have also been observed to change axially and span-wise, supporting the 3D flow. The findings show that as the duct direction deviates from a upward position, the warm air with much lower flow intensity is trapped inside the cavity towards downhill and the conduction mode takes control of the heat transfer. The circulation becomes more vigorous when the entrance is higher up, which has the reverse impact on the flow within the floor hollow, where the surface temperature drops.

Under accident-like circumstances, Wataruat 2008 tested on full scale casks Made up of concrete for heat withdrawal. A concrete-filled steel cask and a reinforced concrete cask are the two options. The air inlets were completely (100%) and partially (50%) closed during the testing. It was found that because the geometry of the airflow area in the reinforced concrete casks varies between the two different types of barrels, the change in the temperature dispersion and the airflow direction differs for every accident scenario. For a 50% obstruction in the entrance region, the concrete barrel's temperature increase is minimal. Reduced airflow rates in RC and CFS barrels are roughly 5% and 22% lower when two of the four air inlets are blocked, respectively. For complete air intake blockage, several flow patterns were observed depending on the geometry of the air exit. When an air outlet is designed in a "down-step" fashion, no air is expelled from the outlet.

A three-dimensional transient CFD model was created by Hung et al. in 2013 to forecast the capacity of spent fuel pools to cool down and eliminate decay heat. For numerical analysis and simulation, the geometry and boundary

conditions of the SFP of Taiwan's second nuclear power plant were utilized as an example. The simulation was performed on the supposition that the external cooling system had failed and that fuels at full-core discharge had been transferred into the SFP. In order to depict the pressure drop of the flow passing through the fuels, the effective permeability is computed using Darcy's law for porous media, which simplified the fuel region.

A dry storage cask with additives and an improved spent nuclear fuel cask were both studied by Bae et al. in 2020. The additive, which is mostly made of borosilicate glass, contains helium. Additionally, chromium is sprayed to the external surface of each addition to improve its solidity and heat-transfer capabilities. The computer model of a dry cask, which was built using 2-D axially symmetric geometry, shows the steady-state thermal results of the system in its solution. A homogenous additive model that had earlier been developed was used to handle additives or small beads. A porous medium was introduced, containing, a heat source, steel frames, additives, helium gas and an artificial alloy. In response to an increase in air velocity, the centreline temperature rapidly decreased until it was close to 490 K. The highest temperature of the system dropped dramatically from 562 K to 472 K. Computational Fluid Dynamics model for non-isothermal mixing ventilation in a standard enclosure was carried out by Kosutova et al. in 2018. A heated floor enclosure was the focus of a through experiment to determine how physical and computational factors affected non-isothermal mixing ventilation. Output from Computational Fluid Dynamic simulations are verified using research data from published studies. It is shown that neither one of the four RANS turbulence models overcomes the others in terms of velocities and temperatures, nor that indoor velocities are more sensitive to changes in physical and computational parameters. The method of predicting mean velocity using various near-wall treatments was shown to have the most influence on the results. Additionally, if the input velocity is reduced by 20% or 40%, the recirculation cell's direction changes from clockwise to anticlockwise. When it came to predicting velocity, the Low - Re k- model fared the best. The outcomes from the first-order patterns were strikingly similar to those from the power-law discretization patterns. For both velocities and temperatures, third-order MUSCL schemes provided results that were very close to those obtained with 2nd-order schemes. While QUICK provided findings for the vertical centre line velocity

that were almost equal to those gained with 2nd-order and MUSCL techniques, Second-order schemes may be preferred, yet the MUSCL and QUICK schemes also displayed highly oscillatory behaviour.

To better understand thermal characteristic under turbulent mixed convection conditions in air-conditioned spaces, Al-Sanea et al., 2012 carried out a CFD study. A slot in the side wall allows air to enter the chamber, and a slot in the floor allows air to exit. It is thermal up there. A careful examination of the dimensions reveals that Re , Gr or Ar (Archimedes number), the room aspect ratio L/H , and the dimensionless heights of the supply and return ports (d/H and t/H), where H is the height of the room, all affect how well the air distribution system performs. By altering each parameter individually, the impacts of Re and Gr or Ar are meticulously investigated for a room with fixed shape. By establishing a standard for distinguishing between forced and mixed convection scenarios, an important Archimedes number of 0.11 is generated. Additional research demonstrates that when Ar is over its critical value, Ar by alone, without regard to the relative significance of Re and Gr , rationally determines Nu_{av} and the heat removal index.

CFD modelling was done by Ji et al. in 2007 for natural ventilation pattern in an enclosure linked to an atrium. The buoyancy-driven ventilation flows in a one-story area that is connected to an atrium are examined using computational fluid dynamics. The atrium, which is higher than the vented room, is warmed by the heat gains from the single-story space, which cause a column of hot air in the atrium and create a ventilated flow. The Computational fluid dynamics simulations either included or did not include the bottom of the atrium ventilation holes, and the outcomes were compared to those anticipated by computational models and small experiments. We looked at the effects of important Computational fluid dynamics modelling challenges that boundary conditions, result restrictions, and mesh dependencies. It was discovered that an ascending vertical thermal plum required the support of a porous region border. Without it, the CFD model failed to fulfil the analytical mode's postulate because the horizontal momentum of the incoming air's impinging action obstructed the development of a vertical plume. The ventilation flow is not constantly improved by a lofty atrium. A small atrium outlet opening could have a greater impact on ventilation flow efficiency than a tall atrium.

In a two-dimensional, partly circular vertical conical area, Nada & Said, 2018 numerically computed thermal convection. A conical cylinder is positioned vertically, and the geometry is schematized for the flow resulting from natural convection in a partial annulus domain. The analytical domain is enclosed between two concentric cones and an isothermal cylinder. It was discovered that conduction heat transfer, which consists of two flow regimes, predominates for lower Rayleigh numbers. At increasing Rayleigh numbers, the rate of heat transfer accelerates and is mostly controlled by convection. One of the more important parameters for flow and temperature fields is the annulus radius ratio. Therefore, for the average heat transfer rate, a low annulus radius ratio is desirable. The rate of heat transfer reduces as the annulus height ratio goes down.

The impact of radiation on natural convection in an annulus with a vertical orientation was investigated by Weng et al. in 1996. The flow field stabilizes and uniformizes as the radiation effect grows. For low aspect ratios, when the conduction to radiation parameter decreases, the temperature at the centre plane increases.

3 AIM AND SCOPE

Experimental studies include the amount of heat generated in each canister for given heat load by carrying out calorimetric studies. Natural circulation pattern and flow rates will be determined at different heat load conditions.

The effect of the ambient conditions and heat load in thermal vault on the cooling will be evaluated. A detailed analysis of the thimble tube with different operating parameters will be carried out based on the air velocity of the stack.

A model will be developed based on the momentum and heat transition equations. The developed model is validated with experimental data. Geometry of thermal vault, gap between thimble tube and ventilation pipe, stack height, heat load and limiting condition of operation are determined by the model.

Experimental and numerical investigation of the studies will provide a detailed insight into type of facility and maximum of activity the facility can accommodate with a limiting condition of 60 °C concrete temperature.

Experiment provides useful information for promising approach of managing heat generated from radioactive canisters in nuclear waste storage by passive cooling. By harnessing natural heat transfer mechanisms, passive cooling systems provide a reliable and energy-efficient solution while ensuring the long-term safety and integrity of the storage facility.

4 EXPERIMENTAL DESCRIPTION

4.1. Experimental setup for Calorimetric Studies

A calorimeter measures the amount of heat transferred during a chemical or physical change. In order to measure the decay heat of VWP Canister, One insulated tank has been made with lid. The tank has provision to measure level by purge method and temperature using two numbers of thermocouples at different elevations. The tank has nozzles inlet (for filling water), outlet (for draining) and over flow. The tank was calibrated with dummy canister, the volume of water required to fill the tank up to over flow is 181 litres and respective level is 1170 mm WC.

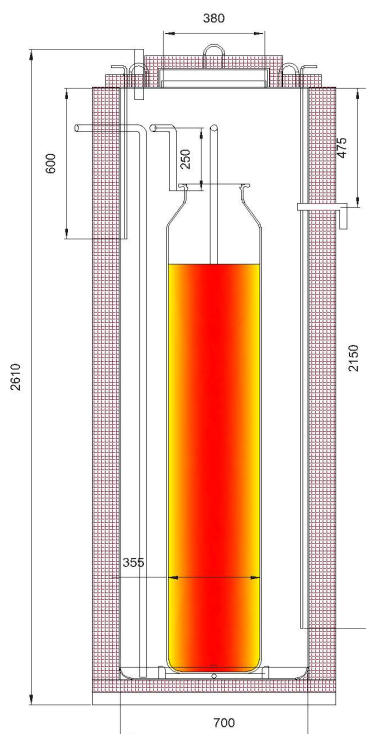


Figure 4.1: Calorimetry setup

The experiment is conducted by filling the tank with a predetermined quantity of water. Approximately, 180 ltr of water is supplied. To ensure the water level, level probes are provided. Ambient temperature is measured using two thermocouples. The canister is lowered into the tank using cell cranes. The lid of the calorimetric tank is closed to minimize thermal losses. The temperature rise, with time is reported.

The heat output of VWP is estimated by $Q = mC_p\Delta T$

Where,

m is mass of water used, C_p is specific heat capacity of water, ΔT is temperature rise,

Heat generation rate $q = Q/\Delta t$

where,

Δt is time required for temperature rise.

Vitrified Waste Storage Facility (VWSF) is intended for interim storage of Vitrified Waste Product (VWP). VWP canisters which are produced from Joule Heated Ceramic Melter are being stored in Vault-1 of VWSF.

The VWP canisters have the decay heat from the vitrified radionuclide present in the canister. The decay heat monitoring is essential as it heavily contributes to the centreline temperatures of the canister and on the de-vitrification of the radionuclide. Hence, the decay heat measurement is necessary for the safe storage of VWP canisters. In order to measure the decay heat, a tank having 100 mm insulation along with level, temperature measurement is installed in the VWSF Hot Cell. 10 no. of canisters were selected with varying activity loaded, which were used for the experiments after sufficient cooling. The tank will be initially filled with measured quantity of 181 lits of DM water, and subsequently the canister is placed in the tank in an immersed condition upto the neck of the canister. The temperature of the DW water were monitored and recorded periodically. The schematic of the experiment is given in Figure 4.2.

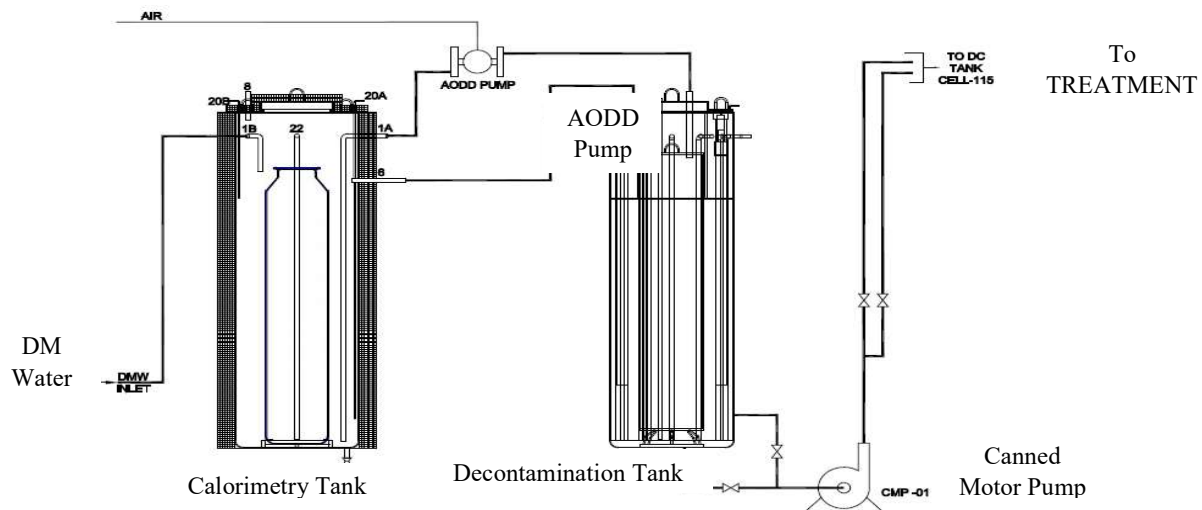


Figure 4.2: Schematic of the Calorimetry at VWSF

4.2. Procedure for Calorimetry Analysis of VWP Canister

- Step 1. Exhaust fan is running and hot cell pressure is maintained better than -2 mmwc.
- Step 2. Ensure that the level and temperature indication of DC tank is available.
- Step 3. Ensure that the level and temperature indication of calorimetry tank is available.
- Step 4. Tank calibration report for DC tank is available.
- Step 5. Tank calibration report of calorimetry tank is available.
- Step 6. Ensure the inlet hose for filling water is in position.
- Step 7. Ensure the tank overflow is lined to cell sump.
- Step 8. Ensure cell sump pump is in working condition.
- Step 9. Ensure the AODD pump is connected from calorimetry tank to DC tank for transferring the liquid after experiment.
- Step 10. Ensure the air connection for AODD pump is available.
- Step 11. Ensure the availability of CMP-01 in hydraulic vault.
- Step 12. Ensure the level of tank TK-234 should be less than 20%
- Step 13. Fill 181 litres of DM water to the calorimetry tank using batching tank. Step 2. Level of the tank should be 860mm.
- Step 14. Transfer canister from cell-114 to hot cell via cell-115 and tunnel.
- Step 15. Place the canister into calorimetry tank.
- Step 16. Ensure the level should be 1770 mmwc and record the temperature.
- Step 17. Continue to record the reading as per the log sheet attached.
- Step 18. After completion of experiment, remove the canister from the Calorimetry tank.
- Step 19. Canister is to be placed in thimble tube by using PPHS.

The activity load and heat generation rate obtained from the VWP are shown in Table 4.1.

Table 4.1: Calorimetric observation

S. No.	VWP No.	Activity Loaded (Ci)	Heat Generation rate kW
1	K-61	156006	0.61
2	K-67	118645	0.50
3	K-10	201763	0.74
4	K-13	201722	0.69
5	K-71	174002	0.65
6	K-25	246061	0.86
7	K-16	207391	0.78
8	K-11	229096	0.82
9	K-09	211930	0.79
10	K-05	191046	0.67



Figure 4.3: Canister loading

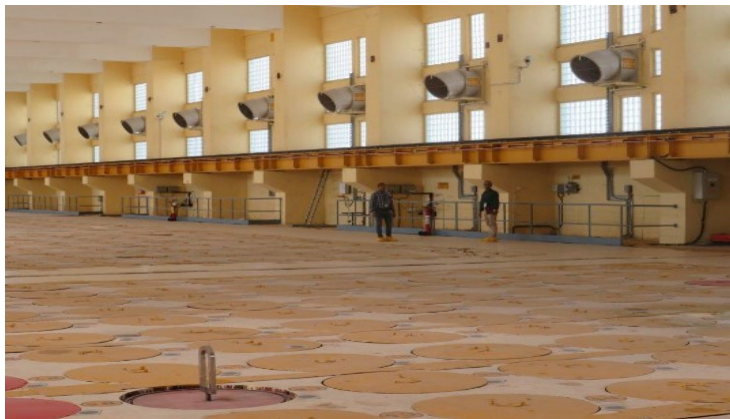


Figure 4.4: Vault area

Table 4.2: Parameters at different heat loading conditions

S. No.	Can. No.	Activity loaded	Heat	Cumu. Heat	Temp (°C)		Velocity (m/s)		Air flow rate	Date of placing
		Ci	kW/can	kW	Ground	Top	Ground	Top	cmh	
1	K-25	246061	0.86	0.86	30	30	2.2	1.5	24334.65	4/8/2023
2	K-11	229096	0.82	1.68	29	29	3	2.4	29636.10	8/8/2023
3	K-10	201763	0.75	2.43	30	30	4.9	1.9	30114.00	10/8/2023
4	K-13	201722	0.70	3.13	30	30	1.5	3.5	31732.20	14/8/2023
5	K-5	191046	0.67	3.8	29	29	3.1	2.8	30592.35	16/8/2023

Table 4.3: Flow Measurement

S. No.	Date	DBT	Air density	Wind Speed VWSF top	Flow rate	Mass flow rate	Wind speed at Stack top	
		°C	kg/m ³	m/s	CMH	kg/s	m/s	km/hr
1	04.08.2023	30	1.16439	1.9	36658.80	11.86	3.91	14.09
2	08.08.2023	30	1.16439	1.2	34905.42	11.29	2.47	8.90
3	10.08.2023	29.5	1.1663	1.8	39680.55	12.86	3.71	13.35
4	14.08.2023	25	1.18391	0.8	30592.35	10.06	1.65	5.93
5	16.08.2023	29	1.16824	1.25	34301.07	11.13	2.57	9.27

4.3. Wind velocity measurement

The mass flow rate through the vault depends on heat generated by VWP, inlet air temperature, and wind velocity at the stack top. The wind speed is measured at a location on the top floor of VWSF vault. The wind speed at the top of stack is determined using exponential Hellmann law. The total heat generated by the three canisters inside the vault is 2.088 kW. The airflow measured for an empty vault is 25833.6 CMH, corresponding to a mass flow rate of 8.356 kg/s, which is attributed to the stack effect.

Hellman Law:

$$v(h) = v_{ref} \left(\frac{h}{h_{ref}} \right)^{\alpha}$$

where,

$v(h)$ is velocity of air at height h

v_{ref} is the reference velocity measured at a known height. In this case, it is the velocity at top of the VWSF building,

and, α is Hellman exponent (0.34 in the present case).

Table 4.4: Parameters used for calculating wind speed

Stack elevation (m)	VWSF top elevation (m)	Ground level (m)	h (m)	h_{ref} (m)
112400	23900	11900	100500	12000

4.4. Development of Two-Dimensional Model

The first step of the two-step analysis has yielded a convergent flow rate. The canisters inside temperature were then measured in detail using COMSOL software. The average heat generation based on radioactive decay of vitrified waste in the vault is used to evaluate the average heat generation rate for all canisters in the model. The 2-D steady state equations of continuity, energy, momentum, and equations of state for air flow are the basis for the model. It is assumed that the entire flow is distributed equally among all the places taken into account. Hence, flow coming out from the inlet duct is simply divided by number of locations to obtain flow rate over each location.

The basic governing equation used to solve the problem is as follows.

Air flow rate due to ΔP is given by

$$\Delta P = k \frac{1}{2} \rho v^2$$

k = coefficient for pressure drop depending on the nature of path and other obstructions

ρ = density of air

v = velocity of air

ΔP = in case of active cooling condition (fan at full capacity + stack effect)

ΔP = in case of passive cooling condition (stack effect)

Heat Balance

(1) Vault

The air flow in the vault is influenced by main stack effect and local stack effect due to rising temp of canisters in passive cooling condition active cooling condition.

The air flow can be found by the above relations from which the outlet temperature of air from the vault can be found by heat balance equation.

$$M_v C_p \frac{dT}{dt} = Q - M_{vf} C_p (T_f - T_o)$$

M_v = Mass of air in vault

M_{vf} = Mass flow rate in vault

T_f = Final temp at vault out

T_o = incoming air temp

$Q = h A \Delta T$

where,

h - heat transfer coefficient

A- Surface Area

ΔT - temperature difference the surface and the surrounding bulk temp

and for forced convection the heat transfer coefficient h can be found by

$$Nu = 0.023 Re^{0.8} Pr^{0.4}$$

$$\text{Nusselt No. } Nu = \frac{hD}{k_a}$$

where

h - heat transfer coefficient

D -equivalent dia.

K_a -thermal conductivity of air

$$\text{Reynold No. } Re = \frac{\rho D v}{\mu}$$

where

μ = coefficient of viscosity,

$$\text{Prandtl No. } Pr = \frac{\mu C_p}{k_a}$$

where C_p = specific heat of air

and due to conduction is

$$Q = kA \frac{\Delta T}{dt}$$

k = conductivity,

dt = thickness

Temperature drop across the centerline temp and the surface temp for cylinder

$$\Delta T = \frac{qr^2}{4k_g}$$

where

q = volumetric heat generation

k_g = thermal conductivity of air

Assumption:

1. The source for the heat inside is the storage cell containing canisters giving out decay heat. Other sources of heat has been ignored. Further for ease of calculation it has been assumed that the storage vault is in parallel to the rest of the plant connected to the stack
2. All the parameters of air and the glass are assumed to be constant.
3. The temperature incoming air is assumed to be 300K. The temperature outside the stack is also assumed to be 300K.

Parameters:

1. Density of air (ρ) = 1 kg/m³
2. Specific Heat of air (C_p) = 1.007 kJ/(kg. °K)
3. Coefficient of volumetric expansion(β)= 3.24x10⁻³/°K
4. Acceleration due to gravity (g) = 10m/s²
5. Conductivity of air (k_a) = 0.02769 W/(m °K)
6. Coefficient of Viscosity of air(μ) = 1.9 x 10⁻⁵ Ns/m²

7. Conductivity of Glass (k_g) = 1.045 W/(m °K)

Final Model

The Transient mathematical model for the air flow through sleeve

$$M_s C_p \frac{dT}{dt} = Q - M_{sf} C_p (T_{sf} - T_0) - U A_s (T_{sf} - T_\infty)$$

And for the room outside the sleeve

$$M_s C_p \frac{dT}{dt} = U A_s (T_{sf} - T_\infty) - M_{sf} C_p (T_c - T_0)$$

Where

M_s - Mass of air in sleeve

M_{sf} - Mass flow rate through sleeve

T_{sf} - Sleeve outlet temp

T_0 - Incoming air temp

U - Over heat transfer coefficient for transfer of heat from canister surface to cell surrounding (is assumed to be 5)

M_c - Mass of air in cell

M_{cf} - Mass flow rate through cell

T_∞ - Temp in the cell

Now,

$$\Delta P_s = \rho g H \beta \Delta T, \quad (\text{Pr. Diff due to Stack effect ie due to } T_s - T_0)$$

$$\Delta P_c = \rho g H \beta \Delta T \quad (\text{Pr. Diff due to cell stack effect i.e due to } T_e - T_0)$$

$$n \Delta P_f + \Delta P_s = k_1 \frac{1}{2} \rho v_0^2$$

$$n \Delta P_f + \Delta P_s + \Delta P_e = k_2 \frac{1}{2} \rho v_e^2$$

where

$n \Delta P_f$ is driving pressure due to fan

$n = 1$ for normal operating condition

$n = 0.4$ during emergency condition

$n = 0$ for black out condition

k_1 = coefficient for pressure drop across the path without the cell

k_2 = coefficient for pressure drop across the path with the storage cell

v_0 = velocity of air through main duct

v_e = velocity of air through the vault.

The problem has been modelled using a software. The model is first calibrated to represent the actual condition of the plant by suitable varying the factor 'k'.

The main cause of air convection that occurs naturally is due to the heat produced within the canisters and, as a result moved to the air passing through the sleeves, and made easier by the draft that was created by the stack. The solution forms the basis of the analysis. Considering the conservation equations for 1-D steady state energy and momentum combined with continuity airflow equation using an iterative approach.

Momentum Equation

$$V \frac{\partial V}{\partial Z} = -\frac{1}{\rho} \frac{\partial P}{\partial Z} - \frac{f_w V^2}{2D_e} - g \cos \theta \quad (1)$$

Continuity Equation

$$\rho AV = \text{constant} \quad (2)$$

Energy Equation

$$Q = hA \Delta T = m C_p \Delta T_f \quad (3)$$

Equation of state

$$\rho = \rho(P, T) \quad (4)$$

After integrating over the flow path, the steady state momentum equation (Eq. (1)) can be given for each section as:

$$P_i - P_{i+1} = \frac{4fL\rho V^2}{2D_e} + C_g \frac{\rho V^2}{2} + \bar{\rho} g L \cos \theta \quad (5)$$

For the Fanning equation, the friction factor (f), is equivalent to 1/4th of the friction factor (f_w), (Darcy–Weisbach equation). Where V is the velocity, Z is the axial distance, f is the fanning friction factor, $\bar{\rho}$ is the average density, and C_g is loss coefficient, which depends upon the geometry. The maximum value of C_g for abrupt expansion is equal to 1.0 and 0.5 for abrupt contraction. θ is the angle from the vertical. The four equations [Eqs. (2) – (5)] are solved to obtain the four variables (V, ρ , T, P). Transfer parameter are related to the main variables mentioned above; for example:

$$f \text{ (friction factor)} = f(V) \quad (6)$$

which can be evaluated

$$f = 0.079 \text{ Re}^{-0.25} \text{ for turbulent flow,}$$

$$\text{Re} < 10^6 \text{ (Blasius equation),}$$

$$f = 16/\text{Re} \text{ for laminar flow.}$$

The buoyancy force generated due to density difference between the inside hot air and outside cold air will stabilise the flow such that it will overcome all

the losses in the vault, duct and the stack. Hence for all the sections Eq. (5) can be modified as:

$$g(Z_{vault} + Z_{stack})\rho_{\infty} = \sum_1^N P_i - P_{i+1} \quad (7)$$

where N is the number of sections, ρ_{∞} is the density of the outside air.

Heat balance equation for the storage unit is given as:

$$Q = \pi R_1^2 q' L = Q_r + Q_c \quad (8)$$

The program is run for heat loads by changing the values of relevant parameters and the values of temperature and mass flow rates are noted down. The observation given in following page is arrived at with an assumption that the sleeve are fully insulated (i.e. overall heat transfer Coefficient $U=0$) and no heat is transferring to the room which will result in maximum temperature at the outlet air.

The vaults are separated by space passages of 1.5 m width. The thimble tubes are arranged on a rectangular pitch. The maximum allowable surface temperature of concrete for the storage vault would be about, and the air inlet temperature is about 30 °C. The inside diameter of the stack at top is 4.0 m and internal diameter of the stack at top of foundation is 7.904 m. The height of stack is 100 m. To remove decay heat, each vault has a separate inlet and output plenum, with little pressure loss and even distribution of incoming air across each thimble tube. The supply air corridor provides inlet air to cool the vault.

This supply corridor is connected to an individual vault by an independent RCC supply plenum. An opening across the width of the vault is provided to uniformly distribute air.

The dynamics of fluid flow, including thermal and flow distribution, are better investigated by CFD. The Navier-Stokes equations, which describe single-phase fluid flows, serve as the foundation for practically all CFD problems. The full potential equations are obtained through further simplification by eliminating components representing vorticity. Advantages of CFD usage is time and cost saving, and reliability. It is recommended to employ CFD when the complexity of the entire system or the geometry prevents the system's behaviour from being calculated using conventional methods. The algorithm involved in CFD studies are shown in Figure4.5.

Table 4.5: VWSF Storage details

Vault no	Vault 1	Vault 2	Vault 3	Vault 4
Size (m)	16.1 x 12.78 x 13	16.1 x 12.78 x 13	16.1 x 12.78 x 13	13.78 x 12.78 x 13
No of locations	136	136	136	117
Capacity	544	544	544	468

Total: 2100 canisters

Table 4.6: Dimensions of Flow Region

Type	Thimble Tube				Ventilation Pipe			
	Length	Inner dia	Outer dia	Thick	Length	Inner dia	Outer dia	Thick
	mm	mm	mm	mm	mm	mm	mm	Mm
Standard	10900	393.7	406.4	6.35	10900	496.92	508	5.54

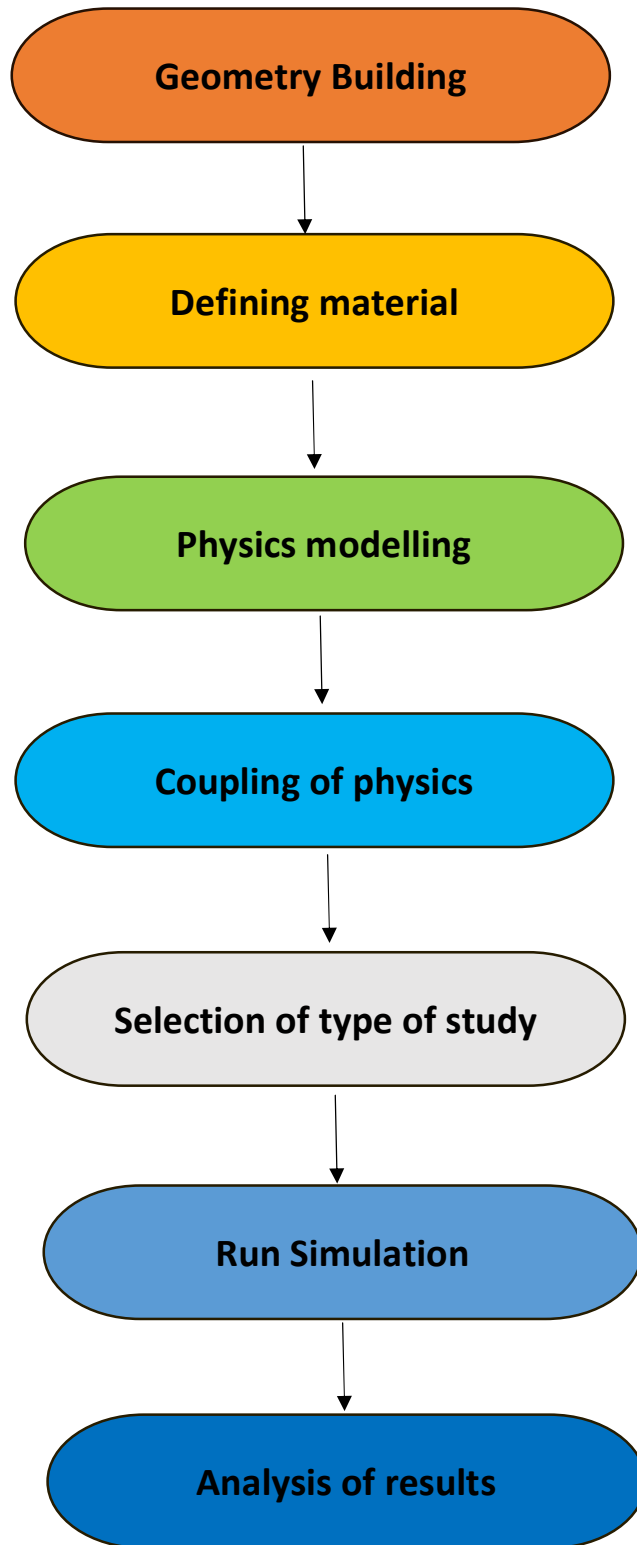


Figure 4.5: Model algorithm

5 RESULTS AND DISCUSSION

The calorimetry experiment is conducted and the results obtained are satisfactory and it indicates that as the Activity loaded in the canister is increasing the decay heat in the canister is increasing. For an average 200000 Ci activity loaded in the canister, the amount of heat generated will be 0.73 kW. The results of the calorimetry experiment are presented in Table and Figure 5.1

Table 5.1: Experimental results of the calorimetry

S. No.	Canister No.	Activity	Temp. Diff.	Heat Generation rate
		Ci	°C	kW/Canister
1	K-61	156006	2.9	0.61
2	K-67	118645	2.4	0.50
3	K-10	201763	3.5	0.74
4	K-13	201722	3.3	0.69
5	K-71	174002	3.1	0.65
6	K-25	246061	4.1	0.86
7	K-16	207391	3.7	0.78
8	K-11	229096	3.9	0.82
9	K-9	211930	3.75	0.79
10	K-5	191046	3.2	0.67

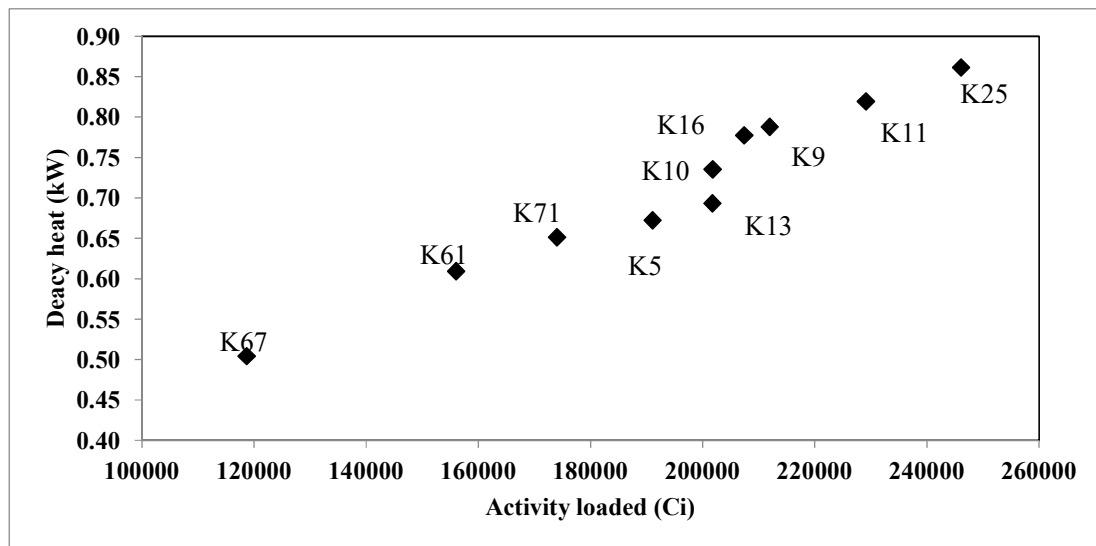


Figure 5.1: Results of calorimetry (Decay heat Vs Activity loaded)

The plot between VWP's heat generation rate and activity is shown in Figure 5.1 It is evident that 0.345 kW of heat is generated corresponding to 1 lack Curie of activity

Figure 4.5 shows relation between wind speed and mass flow rate of cooling air. The mass flow rate at no – load condition is 8.694 kg/s, as evident from the Figure 5.2.

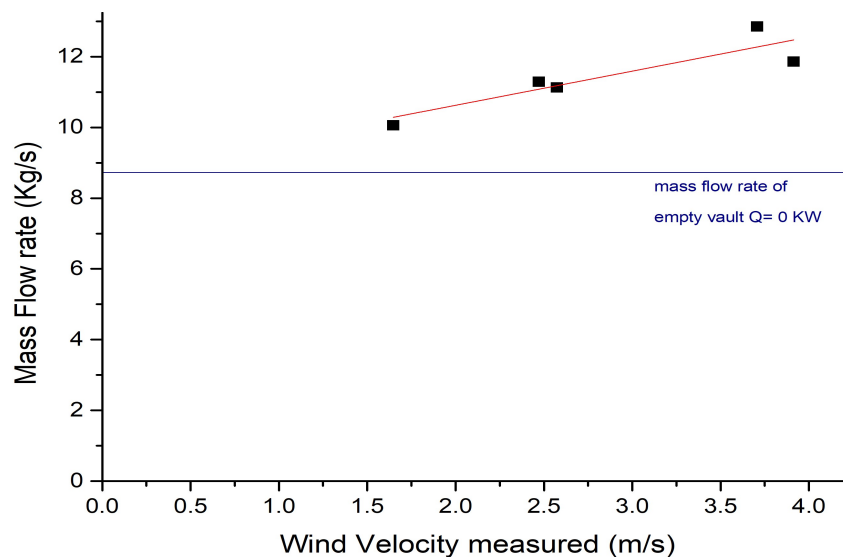


Figure 5.2: Relation between mass flow rate and wind speed for $Q = 2 \text{ kW}$

Since the present study is aimed at demonstration of passive cooling condition under natural draft, the problem adopted is to calculate the airflow rates under the above condition and to obtain the required temperature based on energy balance for corresponding condition.

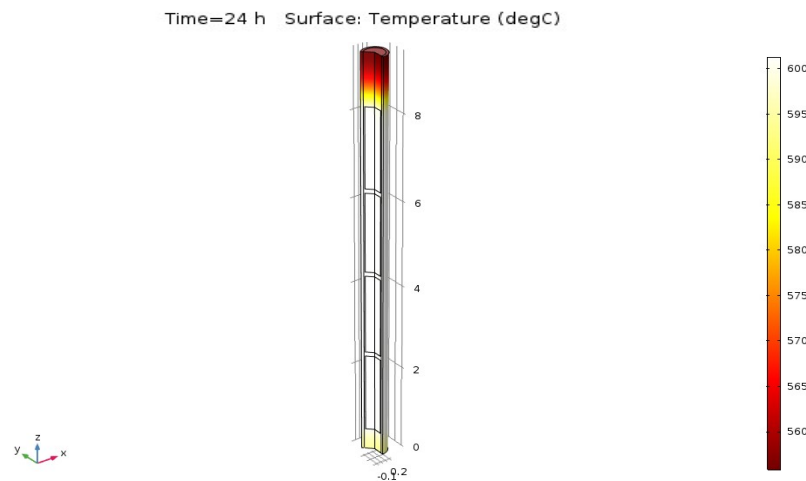


Figure 5.3: Glass temperature profile without cooling

The temperature profile in the thimble tube and ventilation pipe with all the four canisters present is shown in Figure 5.3. In this, the temperature profile is calculated for the canisters with cooling.

The temperature profile with subsequent addition of canisters from one to four is shown in Figure 5.4, 5.6, 5.8 & 5.10. The corresponding velocity profiles are given in Figure 5.5, 5.7, 5.9 & 5.11.

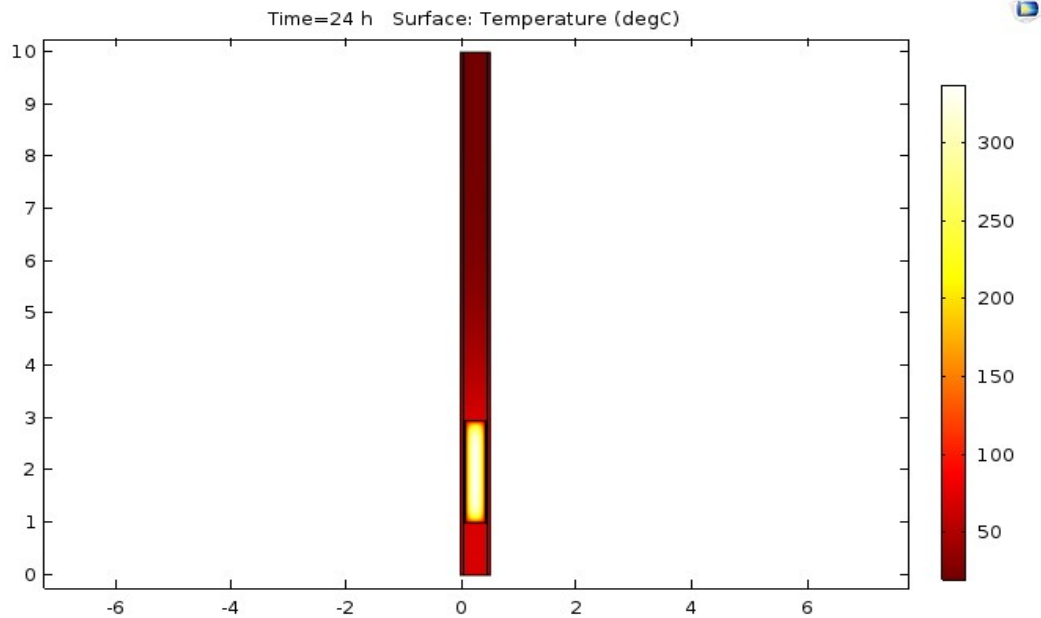


Figure 5.4: Temperature profile for 2 kW decay heat with passive cooling

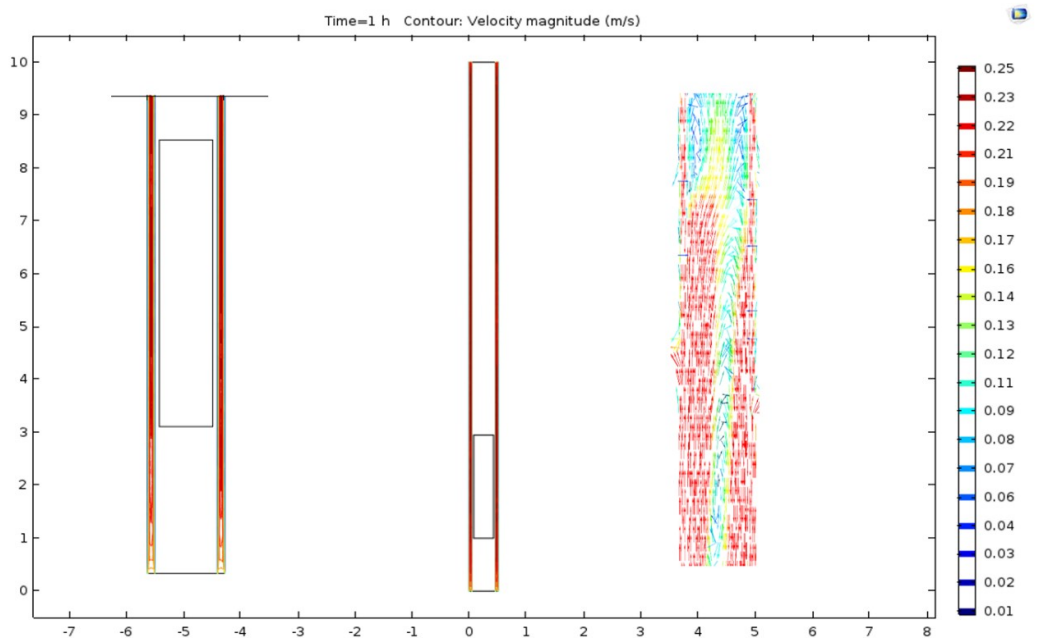


Figure 5.5: Velocity profile across ventilation pipe for 2 kW decay heat

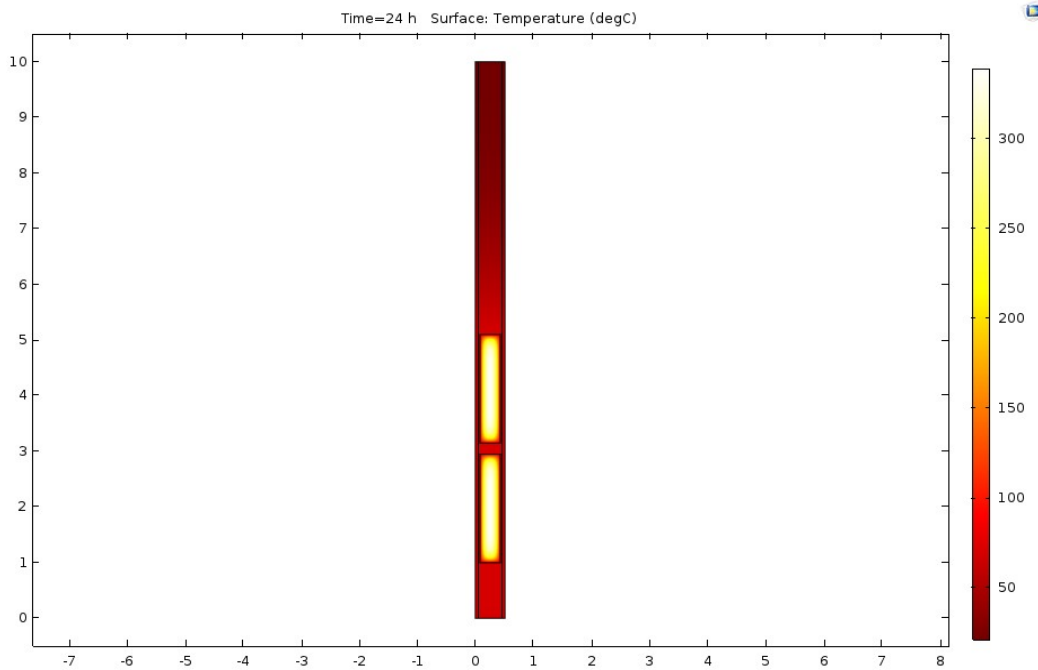


Figure 5.6: Temperature profile for 4 kW decay heat with passive cooling

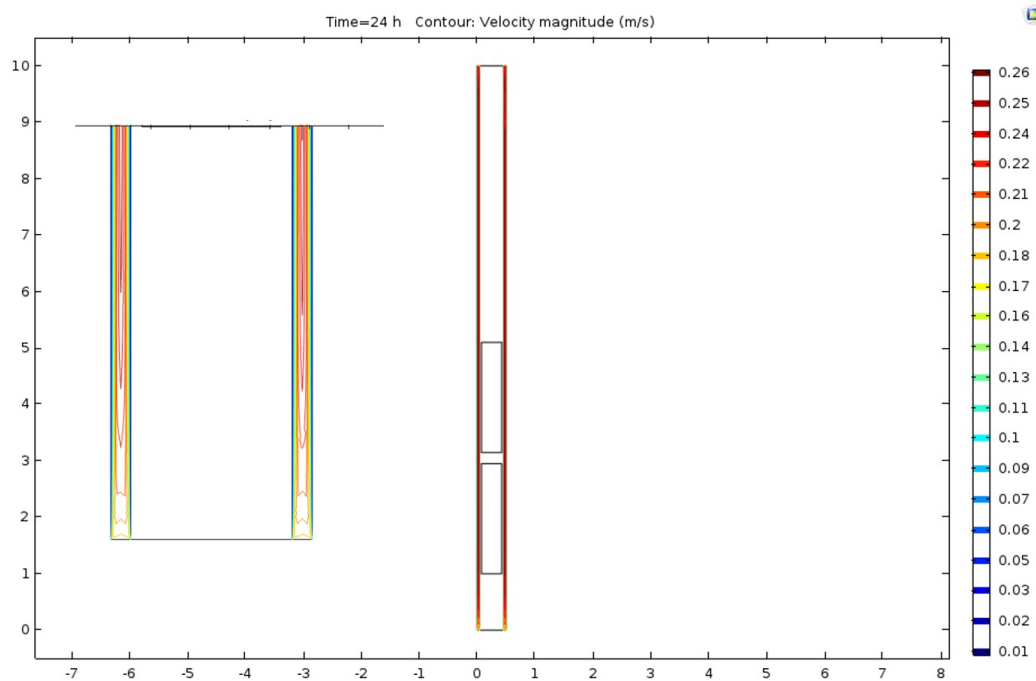


Figure 5.7: Velocity profile across ventilation pipe for 4 kW decay heat

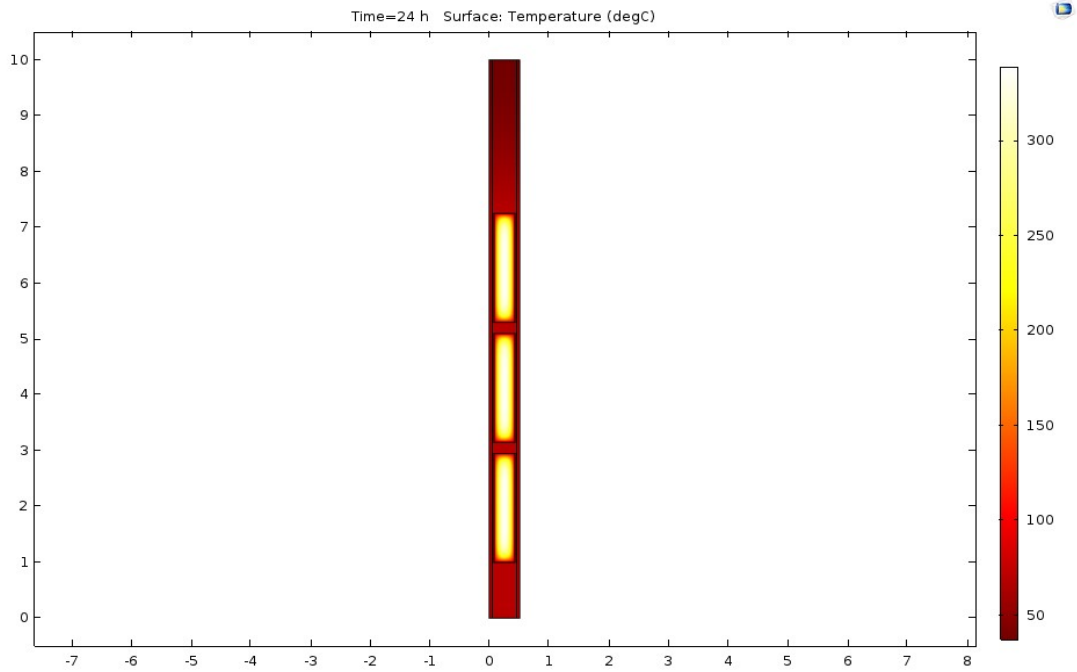


Figure 5.8: Temperature profile for 6 kW decay heat with passive cooling

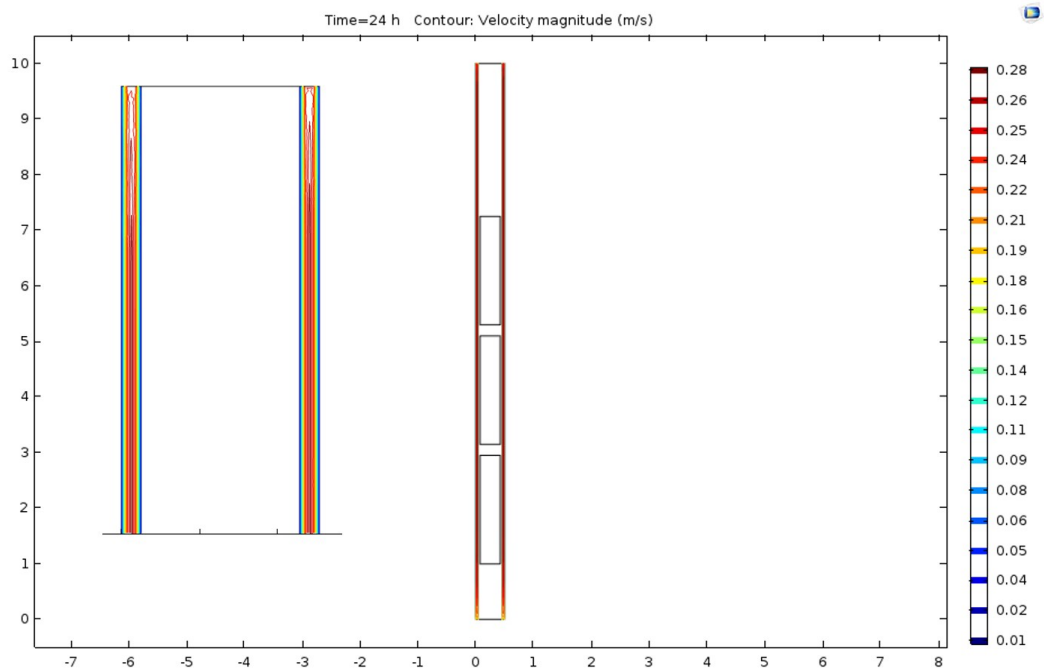


Figure 5.9: Velocity profile across ventilation pipe for 6 kW decay heat

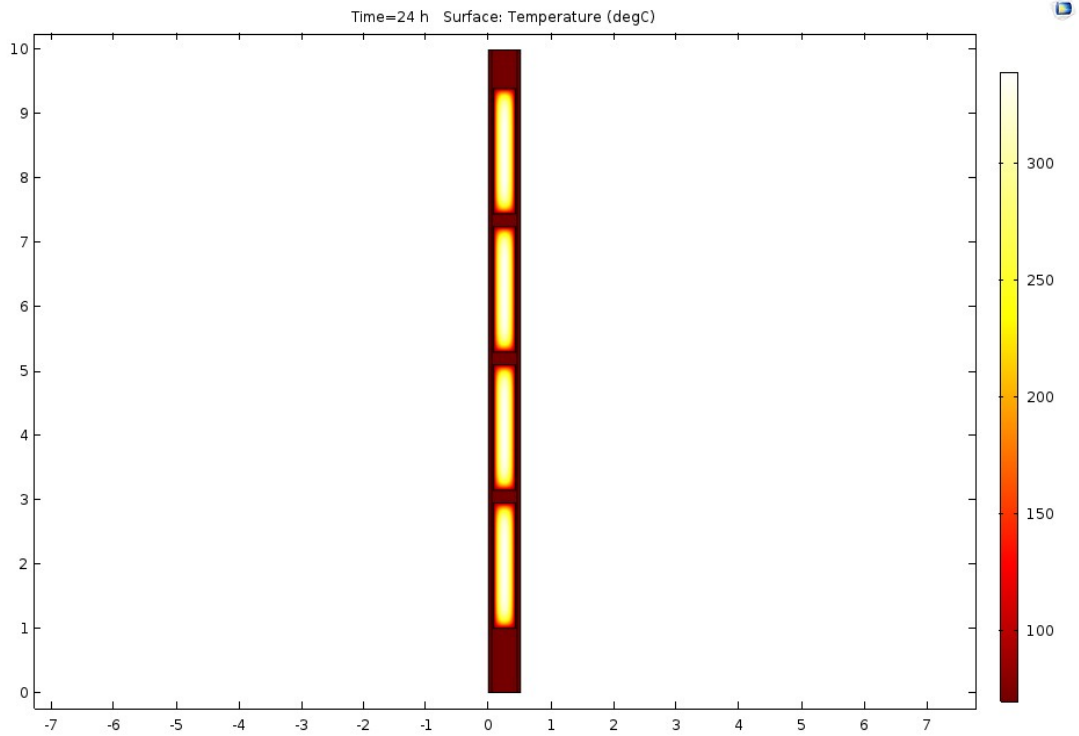


Figure 5.10: Temperature profile for 8 kW decay heat with passive cooling

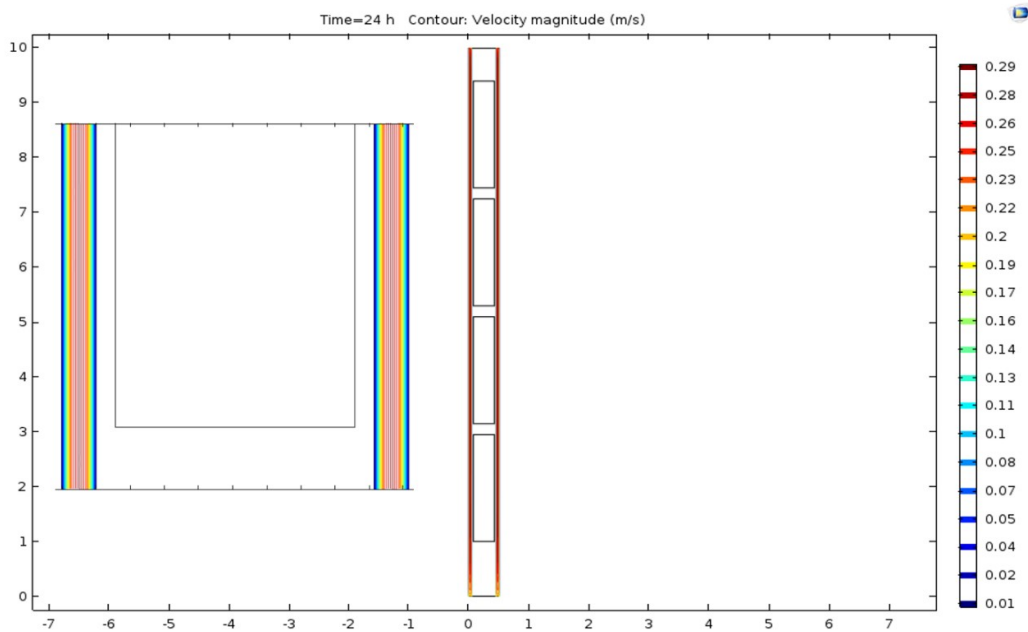


Figure 5.11: Velocity profile across ventilation pipe for 8 kW decay heat

6 SUMMARY AND CONCLUSIONS

Thermal analysis of passive cooling system has been implemented using a parametric analysis, and parameters such as wind flow, heat loading have been chosen. Plenum height, duct dimensions, stack dimensions air gap between canister and thimble tube, annulus flow gap and number of canisters per locations are fixed. The canister centreline temperature is decreases with in the limit by the natural draft. The higher heat load condition increases the flow velocity thus increase in heat transfer coefficient and reduction of canister centreline temperature but it adds the higher air temperature due to reduction in the mass flow rate due to higher pressure drop in the annulus.

The experiment offers important data for a possible method of passive cooling to control heat generated by radioactive canisters in nuclear waste storage. Passive cooling systems offer a dependable and energy-efficient option while preserving the long-term security and integrity of the storage facility by utilizing natural heat transfer mechanisms.

REFERENCES

- [1] Abdullah, M. O., Mikić, F. A., & Lam, C. Y. (2006). Drying performance and thermal transient study with solar radiation supplemented by forced-ventilation. *International Journal of Thermal Sciences*, 45(10), 1027–1034. <https://doi.org/10.1016/j.ijthermalsci.2006.01.013>
- [2] Al-Sanea, S. A., Zedan, M. F., & Al-Harbi, M. B. (2012). Heat transfer characteristics in air-conditioned rooms using mixing air-distribution system under mixed convection conditions. *International Journal of Thermal Sciences*, 59, 247–259. <https://doi.org/10.1016/j.ijthermalsci.2012.04.023>
- [3] Alyokhina, S. (2018). Thermal analysis of certain accident conditions of dry spent nuclear fuel storage. *Nuclear Engineering and Technology*, 50(5), 717–723. <https://doi.org/10.1016/j.net.2018.03.002>
- [4] Bae, J., Bean, R., & Abboud, R. (2020). CFD analysis of a dry storage cask with advanced spent nuclear fuel cask additives. *Annals of Nuclear Energy*, 145. <https://doi.org/10.1016/j.anucene.2020.107610>
- [5] Cárdenas, V., Treviño, C., Rosas, I. Y., & Martínez-Suástegui, L. (2019). Experimental study of buoyancy and inclination effects on transient mixed convection heat transfer in a channel with two symmetric open cubic cavities with prescribed heat flux. *International Journal of Thermal Sciences*, 140, 71–86. <https://doi.org/10.1016/j.ijthermalsci.2019.02.024>
- [6] Chen, H. J., Moshfegh, B., & Cehlin, M. (2012). Numerical investigation of the flow behavior of an isothermal impinging jet in a room. *Building and Environment*, 49(1), 154–166. <https://doi.org/10.1016/j.buildenv.2011.09.027>
- [7] D. B. Mehta, Gattu Suneel, D. Anirudhha, Sangam Gupta and J. K. Gayen, INDIAN EXPERIENCE Vitrification of High Level Liquid Waste.
- [8] Dominguez Espinosa, F. A., & Glicksman, L. R. (2017). Determining thermal stratification in rooms with high supply momentum. *Building and Environment*, 112, 99–114. <https://doi.org/10.1016/j.buildenv.2016.11.016>
- [9] Hung, T. C., Dhir, V. K., Pei, B. S., Chen, Y. S., & Tsai, F. P. (2013). The development of a three-dimensional transient CFD model for predicting cooling ability of spent fuel pools. *Applied Thermal Engineering*, 50(1), 496–504. <https://doi.org/10.1016/j.applthermaleng.2012.06.042>

- [10] Jiao, Z., Yuan, S., Ji, C., Mannan, M. S., & Wang, Q. (2019). Optimization of dilution ventilation layout design in confined environments using Computational Fluid Dynamics (CFD). *Journal of Loss Prevention in the Process Industries*, 60, 195–202. <https://doi.org/10.1016/j.jlp.2019.05.002>
- [11] Ji, Y., Cook, M. J., & Hanby, V. (2007). CFD modelling of natural displacement ventilation in an enclosure connected to an atrium. *Building and Environment*, 42(3), 1158–1172. <https://doi.org/10.1016/j.buildenv.2005.11.002>
- [12] Kim, H. M., No, H. C., Bang, K. S., Seo, K. S., & Lee, S. H. (2014). Development of scaling laws of heat removal and CFD assessment in concrete cask air path. *Nuclear Engineering and Design*, 278, 7–16. <https://doi.org/10.1016/j.nucengdes.2014.06.015>
- [13] Kosutova, K., van Hooff, T., & Blocken, B. (2018). CFD simulation of non-isothermal mixing ventilation in a generic enclosure: Impact of computational and physical parameters. *International Journal of Thermal Sciences*, 129, 343–357. <https://doi.org/10.1016/j.ijthermalsci.2018.03.001>
- [14] Koufi, L., Younsi, Z., Cherif, Y., & Naji, H. (2017). Numerical investigation of turbulent mixed convection in an open cavity: Effect of inlet and outlet openings. *International Journal of Thermal Sciences*, 116, 103–117. <https://doi.org/10.1016/j.ijthermalsci.2017.02.007>
- [15] Kuznik, F., Rusaouën, G., & Brau, J. (2007). Experimental and numerical study of a full scale ventilated enclosure: Comparison of four two equations closure turbulence models. *Building and Environment*, 42(3), 1043–1053. <https://doi.org/10.1016/j.buildenv.2005.11.024>
- [16] Mei, S. J., Hu, J. T., Liu, D., Zhao, F. Y., & Wang, H. Q. (2018). Thermal buoyancy driven flows inside the industrial buildings primarily ventilated by the mechanical fans: Local facilitation and infiltration. *Energy and Buildings*, 175, 87–101. <https://doi.org/10.1016/j.enbuild.2018.07.014>
- [17] Mishra, V. K., Panda, S. K., Sen, B., Maiya, M. P., & Rao, B. P. C. (2022). Numerical analysis of forced convection heat transfer in a nuclear fuel storage vault. *International Journal of Thermal Sciences*, 173. <https://doi.org/10.1016/j.ijthermalsci.2021.107429>
- [18] Nada, S. A., & Said, M. A. (2018). Solutions of thermal performance problems of installing AC outdoor units in buildings light wells using mechanical ventilations.

Applied Thermal Engineering, 131, 295–310.
<https://doi.org/10.1016/j.applthermaleng.2017.12.016>

- [19] Tian, L., Lin, Z., & Wang, Q. (2011). Experimental investigation of thermal and ventilation performances of stratum ventilation. *Building and Environment*, 46(6), 1309–1320. <https://doi.org/10.1016/j.buildenv.2011.01.002>
- [20] Verma, V., Ghosh, A. K., & Kushwaha, H. S. (2002). Simulated model studies for solid waste storage surveillance facility. In *Nuclear Engineering and Design* (Vol. 211). www.elsevier.com/locate/nucengdes
- [21] Verma, V., Ingh, R. K. S., & Vaze, K. K. (n.d.). Thermal analysis of VWSB-IP1 at Tarapur. www.hanser-elibrary.com
- [22] Wang, J., Liu, X., Chen, S., Jiang, H., Fang, G., Chen, W., & Deng, S. (2019). Reduced-scale model study on cable heat dissipation and airflow distribution of power cabins. *Applied Thermal Engineering*, 160. <https://doi.org/10.1016/j.applthermaleng.2019.114068>
- [23] Wang, M., & Chen, Q. (2009). Assessment of various turbulence models for transitional flows in an enclosed environment (RP-1271). *HVAC and R Research*, 15(6), 1099–1119. <https://doi.org/10.1080/10789669.2009.10390881>
- [24] Warren L. McCabe, Julian C. Smith, Peter Harriot; *Unit Operations of Chemical Engineering*, Seventh Edition, McGraw Hill.
- [24] Wataru, M., Takeda, H., Shirai, K., & Saegusa, T. (2008). Heat removal verification tests of full-scale concrete casks under accident conditions. *Nuclear Engineering and Design*, 238(5), 1206–1212. <https://doi.org/10.1016/j.nucengdes.2007.03.035>
- [25] Weng, L. C., Chu, H. S., Weng, L.-C., & Chu, H.-S. (1996). Combined natural convection radiation in a vertical annulus. In *Heat and Mass Transfer* (Vol. 31). Springer-Verlag.
- [26] Wiriyasart, S., & Naphon, P. (2019). Numerical study on air ventilation in the workshop room with multiple heat sources. *Case Studies in Thermal Engineering*, 13. <https://doi.org/10.1016/j.csite.2019.100405>
- [27] Yoo, S. H., No, H. C., Kim, H. M., & Lee, E. H. (2010). Full-scope simulation of a dry storage cask using computational fluid dynamics. *Nuclear Engineering and Design*, 240(12), 4111–4122. <https://doi.org/10.1016/j.nucengdes.2010.08.009>

ORIGINALITY REPORT

5%

SIMILARITY INDEX

1%

INTERNET SOURCES

4%

PUBLICATIONS

1%

STUDENT PAPERS

PRIMARY SOURCES

1

P K WATTAL. "Indian programme on radioactive waste management", Sadhana, 2013

Publication

1%

2

Green Energy and Technology, 2013.

Publication

1%

3

Tzu-Chen Hung, Vijay K. Dhir, Bau-Shei Pei, Yen-Shu Chen, Fengjee P. Tsai. "The development of a three-dimensional transient CFD model for predicting cooling ability of spent fuel pools", Applied Thermal Engineering, 2013

Publication

<1%

4

coek.info

Internet Source

<1%

5

V. Verma, R. K. Singh, K. K. Vaze. "Thermal analysis of VWSB-IP1 at Tarapur", Kerntechnik, 2013

Publication

<1%

6

"Nuclear Fuel Cycle", Springer Science and Business Media LLC, 2023

<1%

7	Submitted to Manipal University Student Paper	<1 %
8	www.science.gov Internet Source	<1 %
9	Submitted to Mount St. Michael's College Student Paper	<1 %
10	Songkran Wiriyasart, Paisarn Naphon. "Numerical study on air ventilation in the workshop room with multiple heat sources", Case Studies in Thermal Engineering, 2019 Publication	<1 %
11	dokumen.pub Internet Source	<1 %
12	Submitted to UCSI University Student Paper	<1 %
13	www.ncbi.nlm.nih.gov Internet Source	<1 %
14	core.ac.uk Internet Source	<1 %
15	www.scribd.com Internet Source	<1 %
16	J. Bae, R. Bean, R. Abboud. "CFD analysis of a dry storage cask with advanced spent nuclear	<1 %

fuel cask additives", Annals of Nuclear Energy,
2020
Publication

17	Raj, K.. "Radioactive waste management practices in India", Nuclear Engineering and Design, 200604	<1 %
----	--	------

Publication

18	Sami A. Al-Sanea, M.F. Zedan, M.B. Al-Harbi. "Heat transfer characteristics in air-conditioned rooms using mixing air-distribution system under mixed convection conditions", International Journal of Thermal Sciences, 2012	<1 %
----	---	------

Publication

Exclude quotes Off
Exclude bibliography On

Exclude matches Off

shanoop p

by Turnitin Official

Submission date: 18-Oct-2023 01:03AM (UTC-0400)

Submission ID: 2199408865

File name: project-report-02.docx (2.13M)

Word count: 8930

Character count: 46356

1 INTRODUCTION

1.1. Radioactive Waste Management

Coal, natural gas, and oil are the three main source of energy used worldwide. Since these resources are not replenishable, they are unable to meet the rising energy demand. In addition, the worry over the greenhouse gas emissions caused by the continued use of fossil fuels necessitates the use of alternative energy sources to lessen the negative effects indicated above. Due to its low carbon footprint, nuclear power can serve as a catalyst for the shift to sustainable energy.

The production of radioactive waste is a side effect of nuclear energy utilisation. These radioactive wastes generate heat, necessitating regular cooling and observation. Immobilisation is the first step in the HLW management approach, which is followed by intermediate storage and disposal. By vitrifying HLW into glass matrix with a Joule-heated ceramic melter, immobilisation is accomplished. Glass and HLW are melted together, and the mixture is then placed into SS304L canisters. Up to 2 kW of decay heat can be produced by a single canister. To lessen the heat produced by radioactive decay, the Vitrified Waste Product (VWP) must be monitored for 30 years.

Vitrified Waste Storage Facility (VWSF) facilitates the storage of VWP. The centre line temperature of VWP should always be maintained below the transition temperature of borosilicate glass that is $\sim 500^{\circ}\text{C}$. The cooling of VWP for a prolonged time is achieved by a natural draft cooling system consisting of a 100 m high stack, which ensures that air flows axially between the ventilation pipe and the thimble tube. Cold air enters the vault via the inlet air corridor and is distributed in a ventilation pipe via the supply plenum at the bottom of the vault. Four canisters can be stored inside the thimble tube. The thimble tube is 11 m long and hung from the top wall of the vault. All storage locations have a lower plenum where air is fed from the inlet duct. This air flows axially via the annular space between the thimble and sleeve. At the top of plenum, the flow that emerges from the top of each storage location is merged before going to the stack. Due to buoyancy, the air flow in the annular space between the ventilation pipe and thimble warms up and rises. Due to the chimney effect, the hot air rises through the stack and keeps the flow through the vault going. The sleeve serves as both a stack and an air flow guide.

1.2. Strategy for HLW management

High level radioactive liquid waste (HLW), which includes the vast majority (99%) of the radioactivity in the whole fuel cycle, is produced during the reprocessing of spent fuel. In addition, after the spent fuel is dissolved for reprocessing, hull waste, or the hollow clad tubes, is formed as solid HLW. The safe treatment of radioactive waste, particularly the HLW, is a key component of public support for nuclear energy. In order to effectively manage HLW, it must be isolated from the biosphere and subjected to long-term monitoring that spans multiple generations. Therefore, in the context of India, the management of HLW includes the following three steps.

1.2.1. Immobilisation of HLW

One of the few nations with expertise in vitrification technology is India. In specially constructed "hot cells" made up of 1.5-meter-thick concrete wall due to the high radiation fields, various operations are remotely carried out. These hot cells have systems and tools for remote handling. Custom-made robots, manipulators, remote welding units and remote surveillance tools are a few of the main remonetisation gadgets. Indian industries, national organizations and academic institutions have actively collaborated on the indigenous development of the remote handling equipment. Glass matrix development for HLW depends on a combination of processing temperatures, specific glass additives, and composition. The achievement maximum loading of waste into glass is constrained by the heat of decay and the solubility of the trash's constituents. Glass-forming additives must adhere to permitted processing temperatures and chemical resistance. These processing temperatures are determined by the radionuclide's particular volatility and the melter material's compatibility with the corrosive environment of molten glass. Due to their restricted solubility or incompatibility with glass composition, the presence of specific chemical species in high level waste, such as thorium, aluminium, fluorine, sulphate, platinum group metals, etc., presents extra challenges for the creation of glass formulations. Utilising cutting-edge analytical tools, the vitrified products are assessed for a number of parameters including chemical durability, radiation stability, melt temperature, homogeneity, waste loading, and thermal stability. The solidified waste form must also satisfy the requirements for short-term and long-term storage before being finally disposed of in a deep geological repository. Figure 1.1 depicts the layout of the ceramic melter

heated by joules principle. Indian programme for managing radioactive waste is vitrifying HLW by using melters.

The VWP is sealed in appropriate containers and overpacks and stored for around 30 years under observation and to allow radioactive decay heat to dissipate. Enough information about the behaviour of the product would be produced over this period of observation. Additionally, it is anticipated that the product's radiation and heat conditions will stabilise to the point where transporting it is practical. A natural flow air conditioning has been created for the vault based on safety and intricate techno-economic factors.

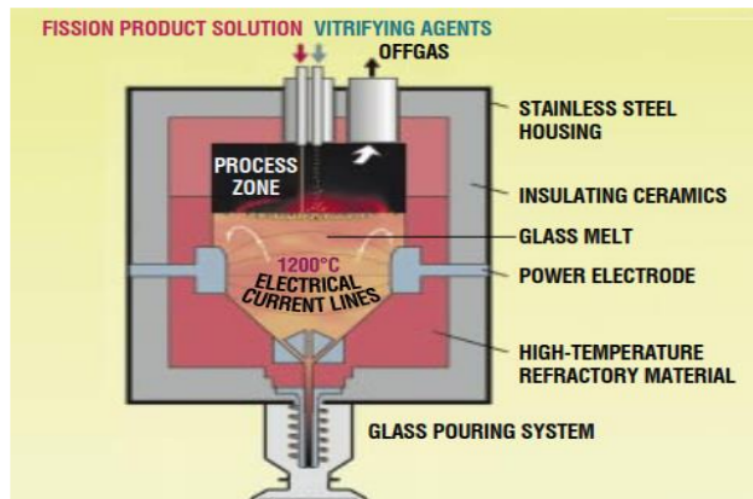


Figure 1.1: Joule Heated Ceramic Melter

1.2.2. Interim storage

For temporary storage, a number of various designs have been taken into account. The use of water ponds as temporary storage for the solidified wastes is merely an expansion of the current method of spent fuel storage. Demineralization and ongoing chemical balancing of the water are required. Both air cooling and water cooling are deemed to be acceptable in terms of safety and thorough technological and economic analysis, however water cooling has a few extra benefits. A superior coolant, water also offers complete activity containment because of its high heat capacity and thermal conductivity compared to air, the canister may fail. However, some drawbacks include corrosion problems and a high make up water need. In India, air-cooled storage is used as a result of a thorough techno-economic analysis.

In India, the passive air-cooling design by natural convection is supplemented by induced draught from the stack, which is independent of people or machinery, is used to store solidified high-level waste. It is using the decay heat and a well-constructed stack to propel air through the storage vault. In practice, the waste heat will raise the air's temperature, which will cause the air to rise as a result of buoyant forces. The driving force brought on by buoyancy is counterbalanced by the friction effect of air passing through the system in order to reach equilibrium for the loading state of each compartment of the vault.

Certain fundamental requirements that, in certain situations, are in conflict must be balanced in the structural facility design. The project site has a higher ground water table, which is projected to surge up to virtually surface level in the monsoon season. Optimum wall thickness is required for the vault to ensure water tightness and constant structural stability against uplift. To minimize heat effects, however, standard concrete must be used for the structure. Only by minimizing the temperature gradients along the concrete wall, which is possible by decreasing the thickness of the vault wall, can this be accomplished. To meet the aforementioned standards, it is essential to isolate the thermal effects from the load-bearing structure as well as any water infiltration. A double vault design is used to accomplish this. The design of the inner storage vault is based on thermal factors, whereas the design of the outer vault is based on biological shielding structural concerns. Additionally, the outer vault would keep groundwater seepage from the local area around the storage structure. Water from seepage should be further processed and activity will be watched for. From the outer vault, specially made bearings support the thermal vault. This enables the thermal vault to slide freely, alleviating the thermal vault of temperature strains.

1.2.3. Final Disposal

International agreement has evolved that deep geological disposal is the best suitable method for permanently separating of vitrified high-level waste from the environment. Remoteness from the surrounding area, the lack of ground water circulation and the capacity to store radionuclides for geologically long times are the three fundamental requirements for a geological formation to be acceptable for the installation of a facility for disposing of radioactive waste. India offers a diverse range of rock types, especially those with a strong capacity to act as a natural barrier to

confine and separate VWP. The most likely possibility for a deep geological deposit could be granites, which make up 20% of the country's overall surface.

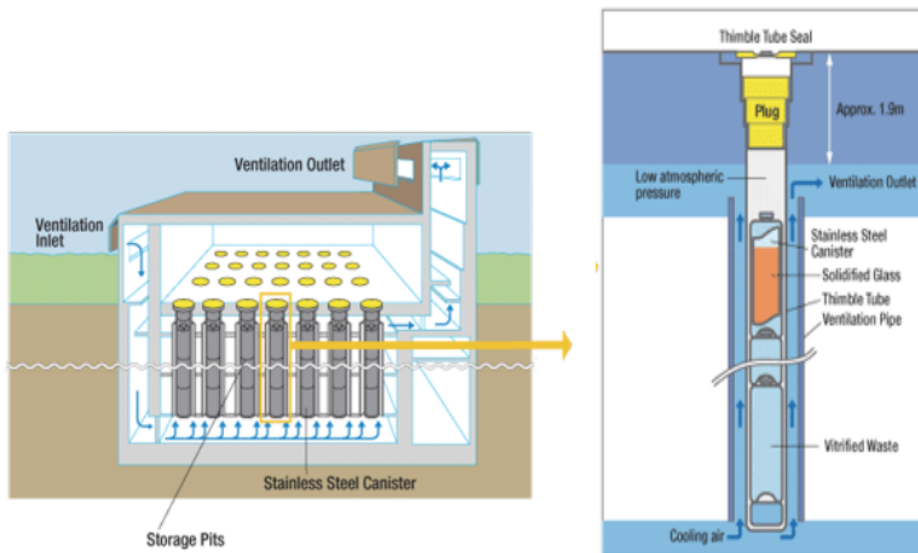
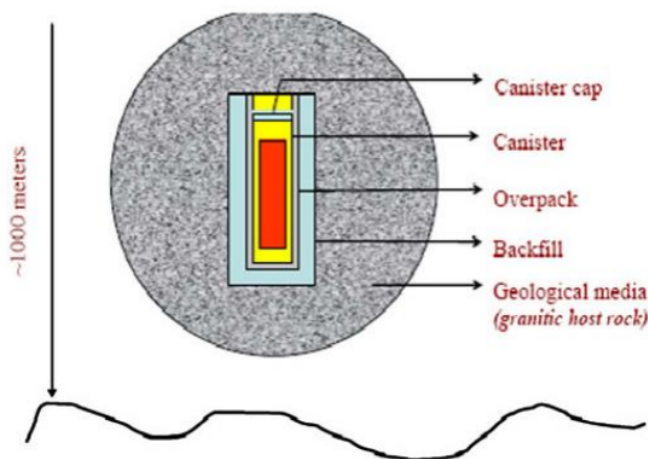


Figure 1.2: interim storage

Even though India won't need a deep geological repository for several decades, work is already being done in the areas of characterization of numerical modelling, natural barriers, conceptual design, and natural analogs of waste forms and repository procedures. To provide more separation, radioactive waste is disposed of using a system of several barriers. The total safety against radioactive migration is achieved through the careful selection of waste form, an appropriate engineered barrier, backfill, and site-specific geo environmental factors. The most crucial elements of the multibarrier strategy used in a deep geological disposal system in hard rocks are backfills and buffers. These are positioned as layers between the host rock and the waste over pack primarily to prevent groundwater from flowing toward the waste form and to slow radionuclide migration into the biosphere in the improbable event that the waste over pack were to be breached and release radionuclides. Bentonitic clays that swell have become more popular options for backfill. For the purpose of evaluating the safety of disposal facilities under different scenarios, model design, execution, and data are crucial. Such safety evaluations are founded on a thorough physical comprehension of the processes impacting the repository and the geological formation, as well as those engaged in the transportation and release of radionuclides. Diagram of the multi-barrier disposal

idea, Figure 1.3. Indian program for managing radioactive waste is reuse and recycle technology development and the desire to efficiently use resources have given rise to possibilities of recycling options that could lighten the load on future generations. Through enhanced plutonium recovery and recycling will considerably decrease the radioactivity of the waste. HLW's long-term radio-toxicity to be less of an environmental burden for nuclear power to continue to grow. The long-lived minor actinides (Am, Np, Cm) and fission products (I-129, Tc-99, etc.) are separated from the radioactive waste in the partitioning and transmutation process, and then they are either rendered non-radioactive or transformed into elements that had a lot shorter half-lives than the original elements. In order to eliminate or decrease radioactive stockpiles, this process may be accomplished in Accelerator Driven Sub-Critical Systems (ADSS) or Integral Fast Reactors (IFR). This long-term approach to managing high-level trash would benefit the environment and the availability of resources. Partitioning of HLW also enables the conditioning of specific radioactive waste streams with the available vitrification technologies using cutting-edge ceramic waste forms like Synroc. Because it is polyphase and polycrystalline, Synroc has an extra benefit for immobilizing high actinide waste loadings. India is working to attain the aforementioned goals through a development program.

Figure 1.3: Deep geological repository



2 LITERATURE SURVEY

2.1. Passive cooling system of heat generating source:

The Vitrified Waste Storage Facility (VWSF) is a facility created to store vitrified high-level waste in retrievable engineered storage canisters. The building is designed to effectively cool product canisters, keeping their temperatures within safe and acceptable limits. VWSF has a 2100 canister storage capacity. Each of the four storage vaults in the VWSF Building can accommodate both common and uncommon canisters.

Using the stack effect, the vault's cooling air is released normally during ventilation without being filtered. Induced ventilation with a High-Efficiency Particulate Absorbing filter (HEPA) is offered if airborne activity is discovered in the exhaust air. Before permanently disposing of the canister in a repository, it is imperative to ensure the integrity of the canister and the product. The heat generated by the disintegration of radioactive nuclides must be dissipated in order to prevent the waste product from melting down under any foreseeable circumstances. Following that, the canister would be qualified for final disposal. The decomposition heat from the canisters would be removed by continuous air movement brought on by natural convection and aided by a stack.

The thimble tubes that house the canisters are held up by a vault roof. There are four canisters piled inside of a thimble tube. Axial airflow is made possible by the air passing around the thimble in a ventilation pipe. Air inlet plenum connects ventilation pipes. An air supply intake tunnel leads air into the vault. It uses an exhaust plenum to leave the vault and enter the stack. Air is channelled axially through a ventilation pipe located around the canisters to provide cooling. If airborne activity is found in the exhaust before it reaches the stack, separate RCC plenums, fans, and filter banks are offered for air filtering. Under normal operating circumstances, there shouldn't be any activity in the exhaust air because air doesn't come into direct touch with the glass product. Table 4.2 displays the precise geometry of ventilation pipes and thimble tubes.

Mishra et al., 2022 carried out a numerical study of the radioactive fuel storage vault with the goal of enhancing ventilation performance by strategically placing interconnected ducts. To decide the layout of numerous connecting ducts, they examined the air flow through the vault. The heat removal efficiency and air

exchange rate through the vault decreases as the aspect ratio of connecting ducts increased.

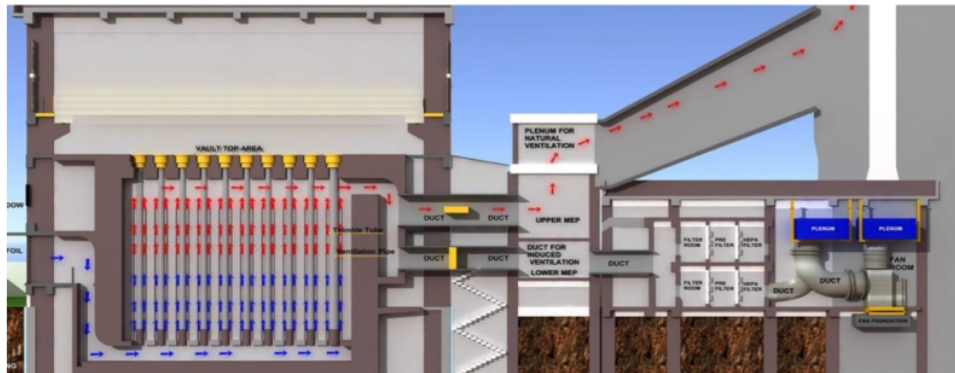


Figure 2.1: Vitrified Waste Storage Facility

Although circular vents give a higher air exchange rate, square ducts are good to eliminating heat. Through the reduction of flow losses (such as frictional losses and losses resulting from duct bends), the smoothing of duct edges enhanced air velocity. By attaching a nozzle to the supply ducts, the performance of the air jet penetration into the fuel pipe was improved. The addition of numerous little air supply nozzles was also shown to be more effective in removing heat than using a big duct. The jet above the heat load helped to better mix the air inside the vault.

CFD modelling was done for solid waste storage facilities by Verma in 2002. For the thermal research, a scaled-down version of such a facility was taken into consideration. Comparisons between theory and experiment have been made. 1-D analysis appears sufficient for the given setup and condition of uniform heating in all the overpacks. The VWP storage vault underwent a numerical examination by Verma et al. in 2013. The parameters, such as duct and stack diameters, plenum height, etc., were studied parametrically. To get a thorough canister temperature profile, CFD code was employed. Outcomes of natural Analysis was done on the convection inside the canister and between the canister and thimble. It was found that the temperature reduces by 20 0C at the canister center line.

M. Wang & Chen conducted studies to investigate the effectiveness of several turbulence models in an enclosed setting in 2009. The survey found that the inflow jet significantly increased space turbulence. The room's air turbulence was lessened by the partition. The study found that while some RANS models did, others did not produce closer estimates for straightforward flows. The LES was the most accurate

and trustworthy model. The DES model under-predicted the turbulent kinetic energy near the wall. If the sub-grid-scale turbulence kinetic energy is taken into account in the DES model, the result can be improved. The complex capabilities of the LES and DES models for handling airflow in enclosed environments are demonstrated in this work.

Chen et al., 2012 examined the RNG k- and SST k-'s propensity to forecast. An extensive experimental examination was carried out, predictions were produced using these RANS models, and the results were used to validate the numerical simulations. Both of the assessed turbulence models can satisfactorily capture the basic qualitative flow features, according to comparisons between experimental data and forecast results. The RNG k- model accurately predicts the maximum velocity decline as the jet reaches the floor, whereas the SST k- model performs well in the region of the impingement zone.

Numerical study was done on a square cavity with a heated floor and two slots—one for the entrance and the other for the outlet—by Koufi et al. in 2017. The locations of the outlet and inlet were adjusted and analysed in order to assess the ventilation effectiveness for temperature dispersion, define the optimal design offering, and determine the required thermal comfort. The governing equations were solved using the finite volume method, and the turbulence closure was performed using the RNG k-turbulence model. The SIMPLEC algorithm was applied to make sure the pressure-velocity correction was made. The layout that maintains thermal comfort the best is one in which the outflow is near the heated floor and the inflow is at the top.

Full-scale experiments and CFD analysis were used by Kuznik et al. in 2007 and he was examined in a mechanically ventilated enclosure, the fields of velocity and temperature. A close-to-the-ceiling axisymmetric jet is created by the ventilation system. The k- SST model, k- model, k- RNG model, and k- realizable model were contrasted in this study. None of the models could accurately predict the cold case. The global temperature and velocity fields, however, only displayed more accurate values in the hot and isothermal instances. Thus, in hot and isothermal conditions, the model can be used to forecast overall comfort as well as the efficiency of thermal ventilation. Even though it does not exactly match the experimental data, the k-model was the most reliable for the cold environment.

CFD models of a dry storage cask were performed by Yoo et al. in 2010, and the outcomes were differentiated with the experimental data. The comparisons have a fair amount of accuracy. The peak guide tube's temperature variation is less than 1 K, which is within the experiment's thermocouples' limit. The size of the basket gap was found to be the most sensitive variable by the analysis.

Kim et al.'s 2014 investigation into heat transport in a concrete cask utilized for PWR spent fuel intermediate storage facilities. To safeguard the fuel cladding, which acts as a barrier for radioactive elements, the decay heat must be appropriately eliminated. To evaluate the design analysis process of the CFD tool, the experimental parameters were established by using half-scale model. The process for building of concrete cask half-scale model for the used dry fuel storage was established using scaling method. The CFD calculations successfully demonstrated a similar temperature rise between the exit and the inlet as well as the efficient maintenance of the exit temperature between the half-scale model and the prototype. The scaling ratios for its exit velocity and the air flow rate were similar with the CFD forecast.

Thermal investigation of accident circumstances in dry spent nuclear fuel storage facilities was done by Alyokhina in 2018. Numerical analysis was done on the flow structure inside ventilated containers. It was discovered what the main mass and heat transport mechanisms were for cooling spent nuclear fuel. Investigations were made into the thermal impacts of accident scenarios including circulated storage containers and channel obstruction. Heat removal in accident conditions is essential but more difficult than simply delivering air through input vents. In order to remove heat, outlet vents in a container's cooling system must be opened. The upper channel has to be opened first and ensure heat dissipation in the case of a complete ventilation duct blockage. For a typical occupied office room, Dominguez Espinosa & Glicksman, 2017 ran CFD simulations in spaces with high supply momentum to examine the impact of ventilation settings and inlet geometry on the air temperature profile in the room. The ventilation factors taken into account in this experiment were heat gain intensity and supply momentum. The height and area of the supply were changed to examine its reliance. The Archimedes number, which contrasts the effects of buoyancy and supply momentum with dimensionless geometric variables, was used to characterize the configuration of the rooms. It was demonstrated that a high Archimedes space might be separated into two region:

one where the temperature increases above the occupants in a warm region with uniform temperature in the other. The buoyancy effects on the flow of the space take precedence over the inertia of the air when the Archimedes number is high. The resulting thermal stratification profile is similar to ventilation displacement. The thermal plums convective effects mix the air as they move upward, creating a layer of warm air that is almost constant in temperature. In a typical Archimedes space, the supply jet mixes the air in the bottom section of the space, particularly near the outlet, making a result of stable temperature in this region. Because the supply jet was poor at mixing the air towards the ceiling, the temperature there was higher than it would have been with a higher Archimedes number. When the Archimedes number is low, the buoyancy effects of the heat sources are defeated by the momentum of the air entering space. Due to the high velocity mixing of the air there, which is extremely close to the outflow, the lower part of the room has an area of roughly equal temperature. For a given Archimedes number, the air temperature dropped in the bottom portion of the room while rising near the ceiling. The vertical mixing in the area benefited from a higher inlet. A bigger intake area results in temperature gradient in the room, with a lower part of the space being cooler and the floor area being warmer. The contour of the temperature profile was not appreciably altered by variable intake regions for samples with low and high Archimedes values. As predicted, it was demonstrated for high Archimedes numbers, the height of the entrance had no impact on the dimensionless temperature. For low Archimedes numbers, however, it had a more significant effect. When the Archimedes number is low, a high inlet lengthens the jet throw, extending the area where the supply air may mix. As a result, the temperature profile appeared to be more consistent, with the exception of regions near the ceiling and floor.

The performance of a dilution ventilation system was examined using CFD by Jiao et al., 2019. Effective dilution ventilation design is crucial for industrial hygiene control. A correctly constructed dilution ventilation system can increase the safety of tight workplaces while preserving comfortable working atmosphere. Due to the fact that the high- air flow area is only concentrated around the cross-sectional area of air intake and outflow, single- inlet ventilation systems only partially reduce pollutant concentrations. The ventilation system's ability to reduce the amount of pollutants in small workplaces is thus diminished. The performance of the ventilation

system is greatly enhanced by two-inlet installation configurations. It permits air to circulate broadly across the small workstation and cover a lot more surface than the standard method, which only has one intake and one outlet. The issue of uni-directional airflow within the small workspace can be successfully avoided by the opposite configuration of the air inlets, which can greatly reduce the pollutant concentration in the downwind area. The buoyancy-driven flows in industrial buildings that are largely ventilated by mechanical fans were examined by Mei et al. in 2018. In order to investigate naturally ventilated flows, an on-site measurement and a CFD model have been employed. The overall cooling needs can be met by the natural ventilation system, but local hotspots are formed at a number of sites because of the vortex airflow structure caused by buoyancy driven flow. To cool just a portion of the building, a cooling equipment using which is located in the same building that it provides cooling with supporting mechanical fans. To determine the ventilation flows, 3D steady-state Reynolds Averaged Navier-Stokes (RANS) equations are employed. On-site measurements were compared to the numerical codes and procedures. Using the average age of the air, natural ventilation's efficacy is evaluated. The average age of air is a mathematical measure of how long it takes a given quantity of air to travel through the system and reach a specific point. In this buoyancy-driven natural ventilation hall, an investigation is conducted into the possibility of employing slabs and mechanical vents to cool the equipment work area. The result of the simulation demonstrate that mechanical vents can successfully reduce air temperature and improve air exchange efficiently in the work area, considering that the ejection direction and velocity are highly related to the well-organized local airflow pattern.

A numerical analysis on air ventilation in the workshop room with different heat sources was carried out by Wiriyasart & Naphon, 2019. A 3-dimensional single-phase turbulent flow model was used in this study to conduct investigations to estimate thermal distribution and air quality in the workplace with diverse heat sources. The experimental findings are using to derive the beginning and boundary conditions. The air temperature and velocity in the workshop room were found to be consistent with the experiment's results. The equipment placed in the workshop and the placement of the ventilation systems have a big impact on the suction force, which transfers air pollution and builds up heat inside the workplace.

A reduced-scale model investigation on the airflow distribution and cable heat dissipation in power cabins was carried out by J. Wang et al. in 2019. On the basis of the Archimedes number, a model at a scale reduction of 1:5 was constructed. The ventilation flow was solved using CFD simulations on the prototype power cabin using the three dimensional steady-state RANS equation and a realizable k-turbulence model. Due to airflow dispersion and cable architecture in the power cabin, the temperature distribution between the cables varies. The amount of turbulence at the air flow's entrance causes a significant drop in temperature.

A fabric drier was created, built, and put through an experimental test by Abdullah et al. in 2006. The results show that forced ventilation adding solar dryers have a higher drying rate than those with natural ventilation. The temperature increases in the solar cum forced-ventilation system is related to the internal heat flux induced by moisture removal and solar radiation, according to early computational calculations based on the Navier-Stokes thermal transient simulation technique. The corresponding experimental data and the numerical simulation are in good agreement. The feasibility of deploying more enormous fans was investigated using parametric research. In especially during the initial drying phases, higher ventilation rates may increase the dryer's performance through an improve temperature rise and in local heat flow.

An experiment was done to investigate the thermal performances of stratum ventilation by Tian et al. in 2011. To combat excessive indoor temperatures, strata ventilation has been advocated. In a strata-ventilated office building, tests on air velocity, temperature, and CO₂ content were done. Four different scenarios—two air volumes of 5.4 or 4.7 air changes per hour and two supply air temperatures (21 or 19 °C) are tested in the experiments. The test room's HVAC system contains a damper that may be changed to control the rate of airflow. An orifice meter used to measure volumetric flow rate. Two no's of thermocouples are used to measure the supply air's temperature. In contrast to the two examples with inlet air temperature of 19 °C, it is found that the two cases with inlet air temperatures of 21 °C function better. In every case, the ventilation efficacy is around 1.5. In order to show inclination and buoyancy impacts on transient mixed convection phenomena in a path with two equal open cubic cavities imposed with heat per unit area, Cárdenas et al., 2019 undertook an experiment. The walls of the holes towards the apertures are exposed to a continual wall heat flux, unlike the other boundary walls of the

channel and the cavities, which are adiabatic and insulated. The experiments are run for a constant Pr 7, a range of buoyancy strengths or modified Richardson numbers from 32 to 300, a channel slope of 0 to 90 degrees, and a Reynolds number of 450 to 1450. The distributions of the nu and mean surface temperatures were averaged. The results demonstrate that, based on the duct direction and buoyancy intensity, the temperature distributions and flow exhibit an undulating pattern and were not equal for the duct centre-line. At high Ri values, the heat transfer rates have also been observed to change axially and span-wise, supporting the 3D flow. The findings show that as the duct direction deviates from a upward position, the warm air with much lower flow intensity is trapped inside the cavity towards downhill and the conduction mode takes control of the heat transfer. The circulation becomes more vigorous when the entrance is higher up, which has the reverse impact on the flow within the floor hollow, where the surface temperature drops.

Under accident-like circumstances, Wataru at 2008 tested on full scale casks Made up of concrete for heat withdrawal. A concrete-filled steel cask and a reinforced concrete cask are the two options. The air inlets were completely (100%) and partially (50%) closed during the testing. It was found that because the geometry of the airflow area in the reinforced concrete casks varies between the two different types of barrels, the change in the temperature dispersion and the airflow direction differs for every accident scenario. For a 50% obstruction in the entrance region, the concrete barrel's temperature increase is minimal. Reduced airflow rates in RC and CFS barrels are roughly 5% and 22% lower when two of the four air inlets are blocked, respectively. For complete air intake blockage, several flow patterns were observed depending on the geometry of the air exit. When an air outlet is designed in a "down-step" fashion, no air is expelled from the outlet.

A three-dimensional transient CFD model was created by Hung et al. in 2013 to forecast the capacity of spent fuel pools to cool down and eliminate decay heat. For numerical analysis and simulation, the geometry and boundary conditions of the SFP of Taiwan's second nuclear power plant were utilized as an example. The simulation was performed on the supposition that the external cooling system had failed and that fuels at full-core discharge had been transferred into the SFP. In order to depict the pressure drop of the flow passing through the fuels, the effective permeability is computed using Darcy's law for porous media, which simplified the fuel region.

A dry storage cask with additives and an improved spent nuclear fuel cask were both studied by Bae et al. in 2020. The additive, which is mostly made of borosilicate glass, contains helium. Additionally, chromium is sprayed to the external surface of each addition to improve its solidity and heat-transfer capabilities. The computer model of a dry cask, which was built using 2-D axially symmetric geometry, shows the steady-state thermal results of the system in its solution. A homogenous additive model that had earlier been developed was used to handle additives or small beads. A porous medium was introduced, containing, a heat source, steel frames, additives, helium gas and an artificial alloy. In response to an increase in air velocity, the centerline temperature rapidly decreased until it was close to 490 K. The highest temperature of the system dropped dramatically from 562 K to 472 K. Computational Fluid Dynamics model for non-isothermal mixing ventilation in a standard enclosure was carried out by Kosutova et al. in 2018. A heated floor enclosure was the focus of a through experiment to determine how physical and computational factors affected non-isothermal mixing ventilation. Output from Computational Fluid Dynamic simulations are verified using research data from published studies. It is shown that neither one of the four RANS turbulence models overcomes the others in terms of velocities and temperatures, nor that indoor velocities are more sensitive to changes in physical and computational parameters. The method of predicting mean velocity using various near-wall treatments was shown to have the most influence on the results. Additionally, if the input velocity is reduced by 20% or 40%, the recirculation cell's direction changes from clockwise to anticlockwise. When it came to predicting velocity, the Low - Re k- model fared the best. The outcomes from the first-order patterns were strikingly similar to those from the power-law discretization patterns. For both velocities and temperatures, third-order MUSCL schemes provided results that were very close to those obtained with 2nd-order schemes. While QUICK provided findings for the vertical centre line velocity that were almost equal to those gained with 2nd-order and MUSCL techniques, Second-order schemes may be preferred, yet the MUSCL and QUICK schemes also displayed highly oscillatory behaviour.

To better understand thermal characteristic under turbulent mixed convection conditions in air-conditioned spaces, Al-Sanea et al., 2012 carried out a CFD study. A slot in the side wall allows air to enter the chamber, and a slot in the floor allows air to exit. It is thermal up there. A careful examination of the dimensions reveals

that Re , Gr or Ar (Archimedes number), the room aspect ratio L/H , and the dimensionless heights of the supply and return ports (d/H and t/H), where H is the height of the room, all affect how well the air distribution system performs. By altering each parameter individually, the impacts of Re and Gr or Ar are meticulously investigated for a room with fixed shape. By establishing a standard for distinguishing between forced and mixed convection scenarios, an important Archimedes number of 0.11 is generated. Additional research demonstrates that when Ar is over its critical value, Ar by alone, without regard to the relative significance of Re and Gr , rationally determines Nu_{av} and the heat removal index.

CFD modelling was done by Ji et al. in 2007 for natural ventilation pattern in an enclosure linked to an atrium. The buoyancy-driven ventilation flows in a one-story area that is connected to an atrium are examined using computational fluid dynamics. The atrium, which is higher than the vented room, is warmed by the heat gains from the single-story space, which cause a column of hot air in the atrium and create a ventilated flow. The Computational fluid dynamics simulations either included or did not include the bottom of the atrium ventilation holes, and the outcomes were compared to those anticipated by computational models and small experiments. We looked at the effects of important Computational fluid dynamics modelling challenges that boundary conditions, result restrictions, and mesh dependencies. It was discovered that an ascending vertical thermal plume required the support of a porous region border. Without it, the CFD model failed to fulfil the analytical mode's postulate because the horizontal momentum of the incoming air's impinging action obstructed the development of a vertical plume. The ventilation flow is not constantly improved by a lofty atrium. A small atrium outlet opening could have a greater impact on ventilation flow efficiency than a tall atrium.

In a two-dimensional, partly circular vertical conical area, Nada & Said, 2018 numerically computed thermal convection. A conical cylinder is positioned vertically, and the geometry is schematized for the flow resulting from natural convection in a partial annulus domain. The analytical domain is enclosed between two concentric cones and an isothermal cylinder. It was discovered that conduction heat transfer, which consists of two flow regimes, predominates for lower Rayleigh numbers. At increasing Rayleigh numbers, the rate of heat transfer accelerates and is mostly controlled by convection. One of the more important parameters for flow and temperature fields is the annulus radius ratio. Therefore, for the average heat

transfer rate, a low annulus radius ratio is desirable. The rate of heat transfer reduces as the annulus height ratio goes down.

The impact of radiation on natural convection in an annulus with a vertical orientation was investigated by Weng et al. in 1996. The flow field stabilizes and uniformizes as the radiation effect grows. For low aspect ratios, when the conduction to radiation parameter decreases, the temperature at the centre plane increases.

3 AIM AND SCOPE

Experimental studies include the amount of heat generated in each canister for given heat loading by carrying out calorimetric studies. Natural circulation pattern and flow rates will be determined at different heat loading conditions.

The effect of the ambient conditions on the cooling will be evaluated. A detailed analysis of the thimble tube with different operating parameters will be carried out based on the air velocity of the stack.

Experiment provides useful information for promising approach of managing heat generated from radioactive canisters in nuclear waste storage by passive cooling. By harnessing natural heat transfer mechanisms, passive cooling systems provide a reliable and energy-efficient solution while ensuring the long-term safety and integrity of the storage facility.

4 EXPERIMENTAL DESCRIPTION

4.1. Experimental setup for Calorimetric Studies

A calorimeter measures the amount of heat transferred during a chemical or physical change. In order to measure the decay heat of VWP Canister, One insulated tank has been made with lid. The tank has provision to measure level by purge method and temperature using two numbers of thermocouples at different elevations. The tank has nozzles inlet (for filling water), outlet (for draining) and over flow. The tank was calibrated with dummy canister, the volume of water required to fill the tank up to over flow is 181 litres and respective level is 1170 mm WC.

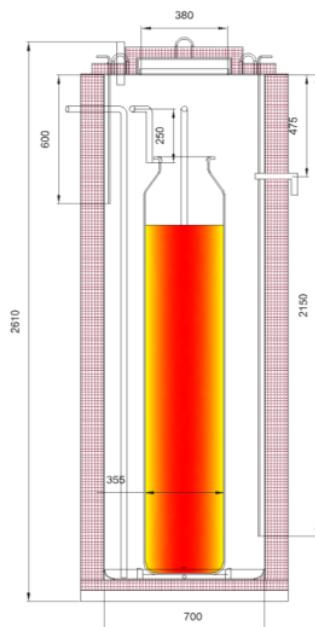


Figure 4.1: Calorimetry setup

The experiment is conducted by filling the tank with a predetermined quantity of water. Approximately, 180 ltr of water is supplied. To ensure the water level, level probes are provided. Ambient temperature is measured using two thermocouples. The canister is lowered into the tank using cell cranes. The lid of the calorimetric tank is closed to minimize thermal losses. The temperature rise, with time is reported.

The heat output of VWP is estimated by $Q = mC_p\Delta T$

Where,

m is mass of water used, C_p is specific heat capacity of water, ΔT is temperature rise, Heat generation rate $q = Q/\Delta t$

where,

Δt is time required for temperature rise.

Vitrified Waste Storage Facility (VWSF) is intended for interim storage of Vitrified Waste Product (VWP). VWP canisters which are produced from Joule Heated Ceramic Melter of are being stored in Vault-1 of VWSF.

The VWP canisters have the decay heat from the vitrified radionuclide present in the canister. The decay heat monitoring is essential as it heavily contributes to the centreline temperatures of the canister and on the de-vitrification of the radionuclide. Hence, the decay heat measurement is necessary for the safe storage of VWP canisters. In order to measure the decay heat, a tank having 100 mm insulation along with level, temperature measurement is installed in the VWSF Hot Cell. 10 no. of canisters were selected with varying activity loaded, which were used for the experiments after sufficient cooling. The tank will be initially filled with measured quantity of 181 lits of DM water, and subsequently the canister is placed in the tank in an immersed condition upto the neck of the canister. The temperature of the DW water were monitored and recorded periodically. The schematic of the experiment is given in Figure 4.2.

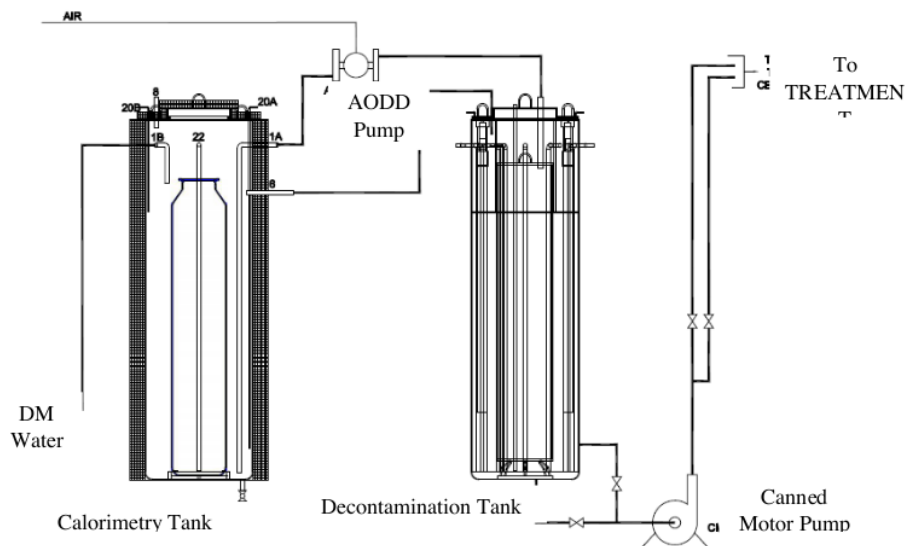


Figure 4.2: Schematic of the Calorimetry at VWSF

4.1 PROCEDURE FOR CALORIMETRY ANALYSIS OF VWP CANISTER

- 4.2.1 Exhaust fan is running and hot cell pressure is maintained better than -2 mmwc.
- 4.2.2 Ensure that the level and temperature indication of DC tank is available.
- 4.2.3 Ensure that the level and temperature indication of calorimetry tank is available.
- 4.2.4 Tank calibration report for DC tank is available.
- 4.2.5 Tank calibration report of calorimetry tank is available.
- 4.2.6 Ensure the inlet hose for filling water is in position.
- 4.2.7 Ensure the tank overflow is lined to cell sump.
- 4.2.8 Ensure cell sump pump is in working condition.
- 4.2.9 Ensure the AODD pump is connected from calorimetry tank to DC tank for transferring the liquid after experiment.
- 4.2.10 Ensure the air connection for AODD pump is available.
- 4.2.11 Ensure the availability of CMP-01 in hydraulic vault.
- 4.2.12 Ensure the level of tank TK-234 should be less than 20%
- 4.2.13 Fill 181 litres of DM water to the calorimetry tank using batching tank.
Level of the tank should be 860 mm.
- 4.2.14 Transfer canister from cell-114 to hot cell via cell-115 and tunnel.
- 4.2.15 Place the canister into calorimetry tank.
- 4.2.16 Ensure the level should be 1770 mmwc and record the temperature.
- 4.2.17 Continue to record the reading as per the log sheet attached.
- 4.2.18 After completion of experiment, remove the canister from the Calorimetry tank.
- 4.2.19 Canister is to be placed in thimble tube by using PPHS.

The activity load and heat generation rate obtained from the VWP are shown in Table 4.1.

Table 4.1: Calorimetric observation

S. No.	VWP No.	Activity Loaded (Ci)	Heat Generation rate kW
1	K-61	156006	0.61
2	K-67	118645	0.50
3	K-10	201763	0.74
4	K-13	201722	0.69
5	K-71	174002	0.65
6	K-25	246061	0.86
7	K-16	207391	0.78
8	K-11	229096	0.82
9	K-09	211930	0.79
10	K-05	191046	0.67



Figure 4.3: Canister loading



Figure 4.4: Vault area

Table 4.2: Parameters at different heat loading conditions

S. No.	Can. No.	Activity loaded Ci	Heat kW/can	Cumulative Heat kW	Temp (°C)		Velocity (m/s)		Air flow rate cmh	Date of placing
					Ground	Top	Ground	Top		
1	K-25	246061	0.86	0.86	30	30	2.2	1.5	24334.65	4/8/2023
2	K-11	229096	0.82	1.68	29	29	3	2.4	29636.10	8/8/2023
3	K-10	201763	0.75	2.43	30	30	4.9	1.9	30114.00	10/8/2023
4	K-13	201722	0.70	3.13	30	30	1.5	3.5	31732.20	14/8/2023
5	K-5	191046	0.67	3.8	29	29	3.1	2.8	30592.35	16/8/2023

Table 4.3: Flow Measurement

S. No.	Date	DBT °C	Air density kg/m ³	Wind Speed VWSF top m/s	Flow rate CMH	Mass flow rate kg/s	Wind speed at Stack top	
							m/s	km/hr
1	18.02.2021	30	1.16439	1.9	36658.80	11.86	3.91	14.09
2	03.03.2021	30	1.16439	1.2	34905.42	11.29	2.47	8.90
3	06.03.2021	29.5	1.1663	1.8	39680.55	12.86	3.71	13.35
4	08.03.2021	25	1.18391	0.8	30592.35	10.06	1.65	5.93
5	09.03.2021	29	1.16824	1.25	34301.07	11.13	2.57	9.27

4.2 Wind velocity measurement

The mass flow rate through the vault depends on heat generated by VWP, inlet air temperature, and wind velocity at the stack top. The wind speed is measured at a location on the top floor of VWSF vault. The wind speed at the top of stack is determined using exponential Hellmann law. The total heat generated by the three canisters inside the vault is 2.088 kW. The airflow measured for an empty vault is 25833.6 CMH, corresponding to a mass flow rate of 8.356 kg/s, which is attributed to the stack effect.

Hellman Law:

$$v(h) = v_{ref} \left(\frac{h}{h_{ref}} \right)^\alpha$$

where,

$v(h)$ is velocity of air at height h

v_{ref} is the reference velocity measured at a known height. In this case, it is the velocity at top of the VWSF building,

and, α is Hellman exponent (0.34 in the present case).

Table 4.4: Parameters used for calculating wind speed

Stack elevation (m)	VWSF top elevation (m)	Ground level (m)	h (m)	h_{ref} (m)
112400	23900	11900	100500	12000

4.2.DEVELOPMENT OF TWO-DIMENSIONAL MODEL

The first step of the two-step analysis has yielded a convergent flow rate. The canisters inside temperature were then measured in detail using COMSOL software. The average heat generation based on radioactive decay of vitrified waste in the vault is used to evaluate the average heat generation rate for all canisters in the model. The 2-D steady state equations of continuity, energy, momentum, and equations of state for air flow are the basis for the model. It is assumed that the entire flow is distributed equally among all the places taken into account. Hence, flow coming out from the inlet duct is simply divided by number of locations to obtain flow rate over each location.

The basic governing equation used to solve the problem is as follows.

Air flow rate due to ΔP is given by

$$\Delta P = k \frac{1}{2} \rho v^2$$

k = coefficient for pressure drop depending on the nature of path and other obstructions

ρ = density of air

v = velocity of air

ΔP = in case of active cooling condition (fan at full capacity + stack effect)

ΔP = in case of passive cooling condition (stack effect)

Heat Balance

(1) Vault

The air flow in the vault is influenced by main stack effect and local stack effect due to rising temp of canisters in passive cooling condition active cooling condition.

The air flow can be found by the above relations from which the outlet temperature of air from the vault can be found by heat balance equation.

$$M_v C_p \frac{dT}{dt} = Q - M_{vf} C_p (T_f - T_o)$$

M_v = Mass of air in vault

M_{vf} = Mass flow rate in vault

T_f = Final temp at vault out

T_o = incoming air temp

$$Q = h A \Delta T$$

where,

h - heat transfer coefficient

A - Surface Area

ΔT - temperature difference the surface and the surrounding bulk temp

and for forced convection the heat transfer coefficient h can be found by

$$Nu = 0.023 Re^{0.8} Pr^{0.4}$$

$$\text{Nusselt No. } Nu = \frac{hD}{k_a}$$

where

h - heat transfer coefficient

D - equivalent dia.

K_a -thermal conductivity of air

$$\text{Reynold No. } Re = \frac{\rho D v}{\mu}$$

where

μ = coefficient of viscosity,

$$\text{Prandtl No. } Pr = \frac{\mu C_p}{k_a}$$

where C_p = specific heat of air

and due to conduction is

$$Q = kA \frac{\Delta T}{dt}$$

k = conductivity,

dt = thickness

Temperature drop across the centerline temp and the surface temp for cylinder

$$\Delta T = \frac{qr^2}{4k_g}$$

where

q = volumetric heat generation

k_g = thermal conductivity of air

Assumption:

1. The source for the heat inside is the storage cell containing canisters giving out decay heat. Other sources of heat has been ignored. Further for ease of calculation it has been assumed that the storage vault is in parallel to the rest of the plant connected to the stack
2. All the parameters of air and the glass are assumed to be constant.
3. The temperature incoming air is assumed to be 300 K. The temperature outside the stack is also assumed to be 300 K.

Parameters:

1. Density of air (ρ) = 1 kg/m³
2. Specific Heat of air (C_p) = 1.007 kJ/(kg. °K)
3. Coefficient of volumetric expansion(β)= 3.24x10⁻³/°K
4. Acceleration due to gravity (g) = 10m/s²
5. Conductivity of air (k_a) = 0.02769 W/(m °K)
6. Coefficient of Viscosity of air(μ) = 1.9 x 10⁻⁵ Ns/m²

7. Conductivity of Glass (k_g) = 1.045 W/(m °K)

Final Model

The Transient mathematical model for the air flow through sleeve

$$M_s C_p \frac{dT}{dt} = Q - M_{sf} C_p (T_{sf} - T_0) - U A_s (T_{sf} - T_\infty)$$

And for the room outside the sleeve

$$M_s C_p \frac{dT}{dt} = U A_s (T_{sf} - T_\infty) - M_{cf} C_p (T_c - T_0)$$

Where

M_s - Mass of air in sleeve

M_{sf} - Mass flow rate through sleeve

T_{sf} - Sleeve outlet temp

T_0 - Incoming air temp

U- Over heat transfer coefficient for transfer of heat from canister surface to cell surrounding (is assumed to be 5)

M_c - Mass of air in cell

M_{cf} - Mass flow rate through cell

T_∞ - Temp in the cell

Now,

$$\Delta P_s = \rho g H \beta \Delta T, \quad (\text{Pr. Diff due to Stack effect ie due to } T_s - T_0)$$

$$\Delta P_e = \rho g H \beta \Delta T \quad (\text{Pr. Diff due to cell stack effect i.e due to } T_e - T_0)$$

$$n \Delta P_f + \Delta P_s = k_1 \frac{1}{2} \rho v_0^2$$

$$n \Delta P_f + \Delta P_s + \Delta P_e = k_2 \frac{1}{2} \rho v_e^2$$

where

$n \Delta P_f$ is driving pressure due to fan

$n=1$ for normal operating condition

$n=0.4$ during emergency condition

$n=0$ for black out condition

k_1 = coefficient for pressure drop across the path without the cell

k_2 = coefficient for pressure drop across the path with the storage cell

v_0 = velocity of air through main duct

v_e = velocity of air through the vault.

The problem has been modelled using a software. The model is first calibrated to represent the actual condition of the plant by suitable varying the factor 'k'.

The program is run for heat loads by changing the values of relevant parameters and the values of temperature and mass flow rates are noted down.

The observation given in following page is arrived at with an assumption that the sleeve are fully insulated (i.e. overall heat transfer Coefficient $U=0$) and no heat is transferring to the room which will result in maximum temperature at the outlet air.

The vaults are separated by space passages of 1.5 m width. The thimble tubes are arranged on a rectangular pitch. The maximum allowable surface temperature of concrete for the storage vault would be about, and the air inlet temperature is about 30 °C. The inside diameter of the stack at top is 4.0 m and internal diameter of the stack at top of foundation is 7.904 m. The height of stack is 100 m. To remove decay heat, each vault has a separate inlet and output plenum, with little pressure loss and even distribution of incoming air across each thimble tube. The supply air corridor provides inlet air to cool the vault.

This supply corridor is connected to an individual vault by an independent RCC supply plenum. An opening across the width of the vault is provided to uniformly distribute air.

The dynamics of fluid flow, including thermal and flow distribution, are better investigated by CFD. The Navier-Stokes equations, which describe single-phase fluid flows, serve as the foundation for practically all CFD problems. The full potential equations are obtained through further simplification by eliminating components representing vorticity. Advantages of CFD usage is time and cost saving, and reliability. It is recommended to employ CFD when the complexity of the entire system or the geometry prevents the system's behaviour from being calculated using conventional methods. The algorithm involved in CFD studies are shown in Figure 4.5.

Table 4.5: VWSF Storage details

Vault no	Vault 1	Vault 2	Vault 3	Vault 4
Size (m)	16.1 x 12.78 x 13	16.1 x 12.78 x 13	16.1 x 12.78 x 13	13.78 x 12.78 x 13
No of locations	136	136	136	117
Capacity	544	544	544	468

Total: 2100 canisters

Table 4.6: Dimensions of Flow Region

Type	Thimble Tube				Ventilation Pipe		
	Length	Inner dia	Outer dia	Thick	Length	Inner dia	Outer dia
	mm	mm	mm	mm	mm	mm	mm
Standard	10900	393.7	406.4	6.35	10900	496.92	508
							5.54

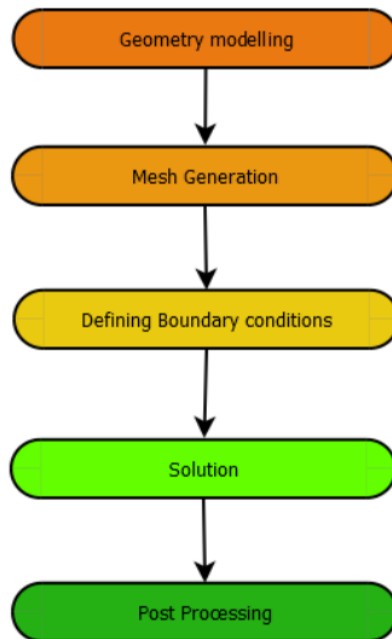


Figure 4.5: Model algorithm

5 RESULTS AND DISCUSSION

The calorimetry experiment is conducted and the results obtained are satisfactory and it indicates that as the Activity loaded in the canister is increasing the decay heat in the canister is increasing. For an average 200000 Ci activity loaded in the canister, the amount of heat generated will be 0.73 kW. The results of the calorimetry experiment are presented in Table and Figure 5.1

Table 5.1: Experimental results of the calorimetry

S. No.	Canister No.	Activity	Temp. Diff.	Heat Generation rate
		Ci	°C	kW/Canister
1	K-61	156006	2.9	0.61
2	K-67	118645	2.4	0.50
3	K-10	201763	3.5	0.74
4	K-13	201722	3.3	0.69
5	K-71	174002	3.1	0.65
6	K-25	246061	4.1	0.86
7	K-16	207391	3.7	0.78
8	K-11	229096	3.9	0.82
9	K-9	211930	3.75	0.79
10	K-5	191046	3.2	0.67

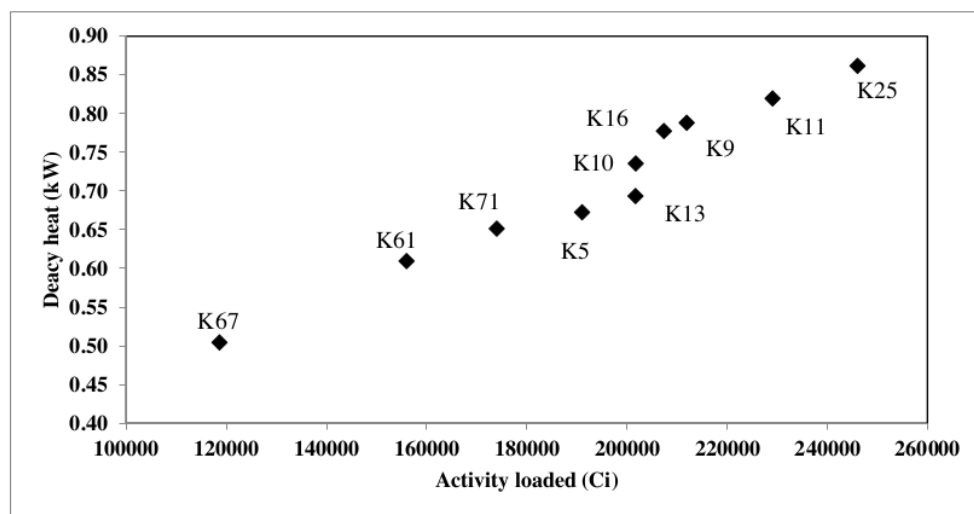


Figure 5.1: Results of calorimetry (Decay heat Vs Activity loaded)

The plot between VWP's heat generation rate and activity is shown in Figure 5.1. It is evident that 0.345 kW of heat is generated corresponding to 1 lack Curie of activity

Figure 4.5 shows relation between wind speed and mass flow rate of cooling air. The mass flow rate at no – load condition is 8.694 kg/s, as evident from the Figure 3.2.

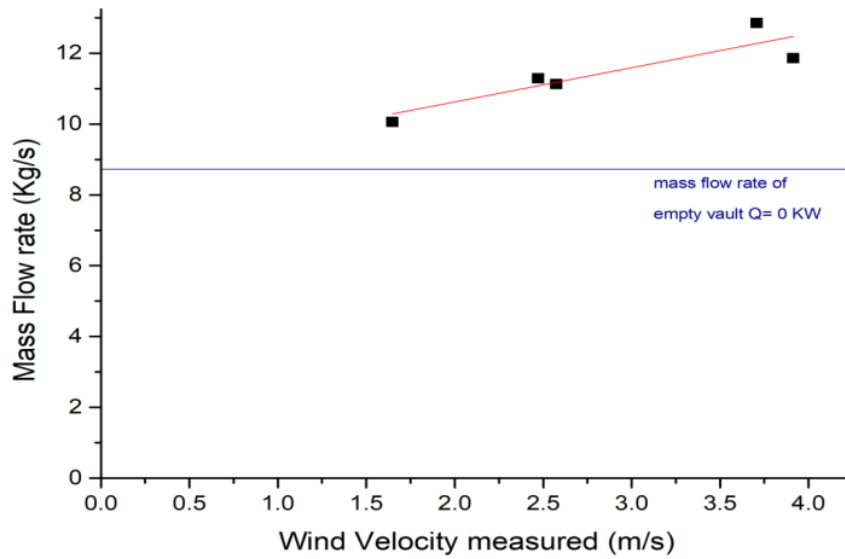


Figure 5.2: Relation between mass flow rate and wind speed for $Q = 2 \text{ kW}$

Since the present study is aimed at the state condition under passive cooling, the problem adopted is to calculate the airflow rates under the above condition and to obtain the required temperature based on energy balance for corresponding condition.

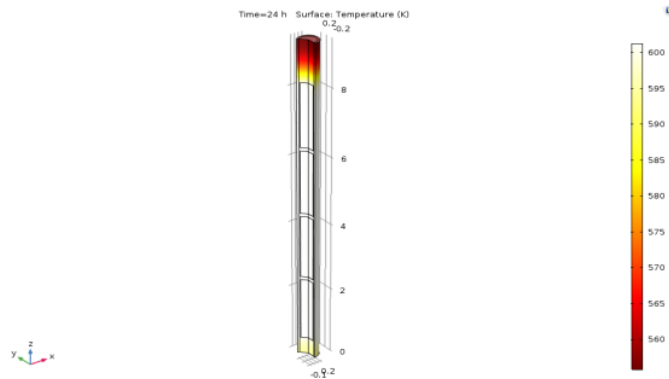


Figure 5.3: Glass temperature profile without cooling

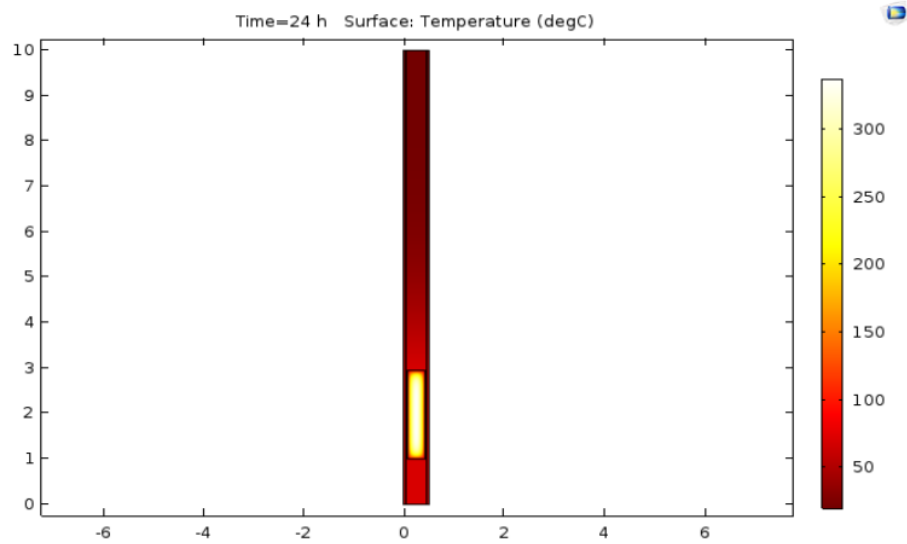


Figure 5.4: Temperature profile for 2 kW decay heat with passive cooling

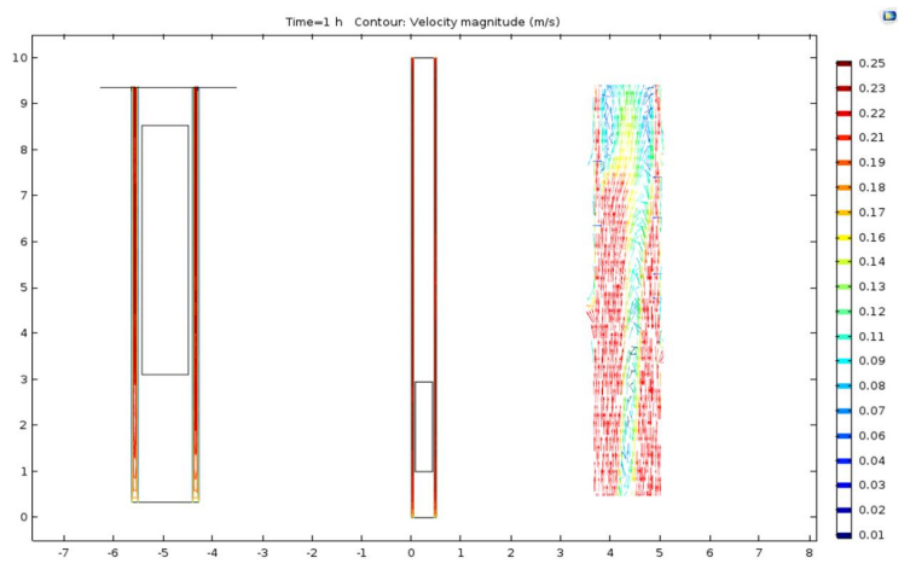


Figure 5.5: Velocity profile across ventilation pipe for 2 kW decay heat

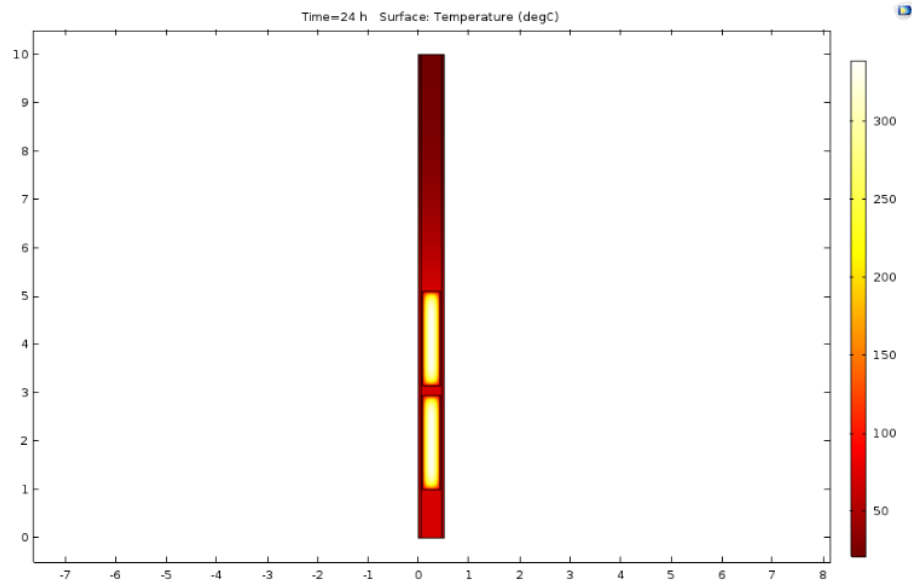


Figure 5.6: Temperature profile for 4 kW decay heat with passive cooling

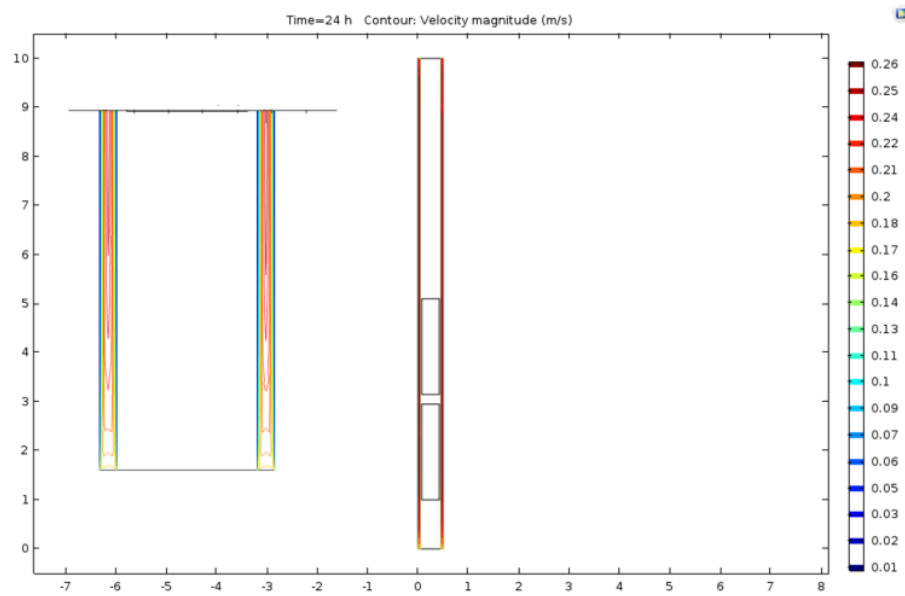


Figure 5.7: Velocity profile across ventilation pipe for 4 kW decay heat

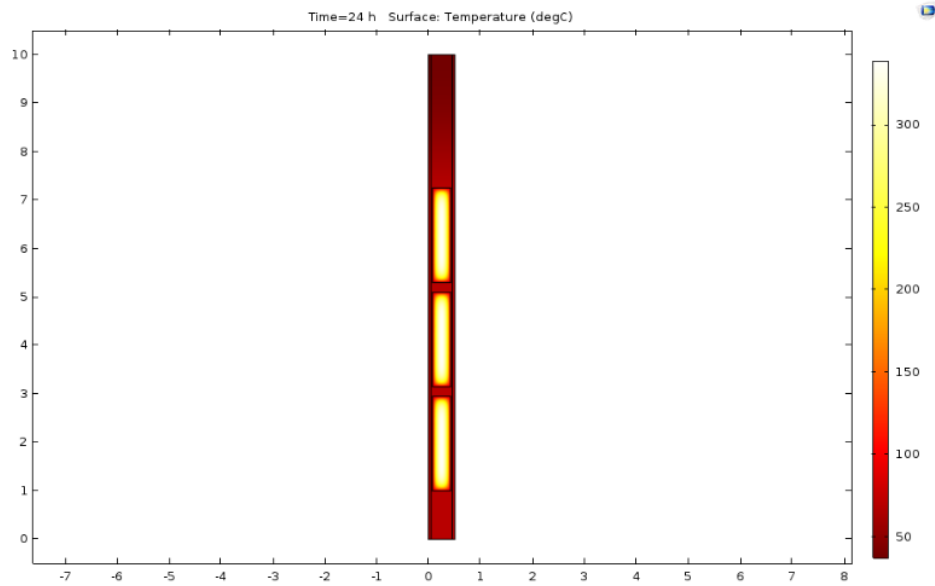


Figure 5.8: Temperature profile for 6 kW decay heat with passive cooling

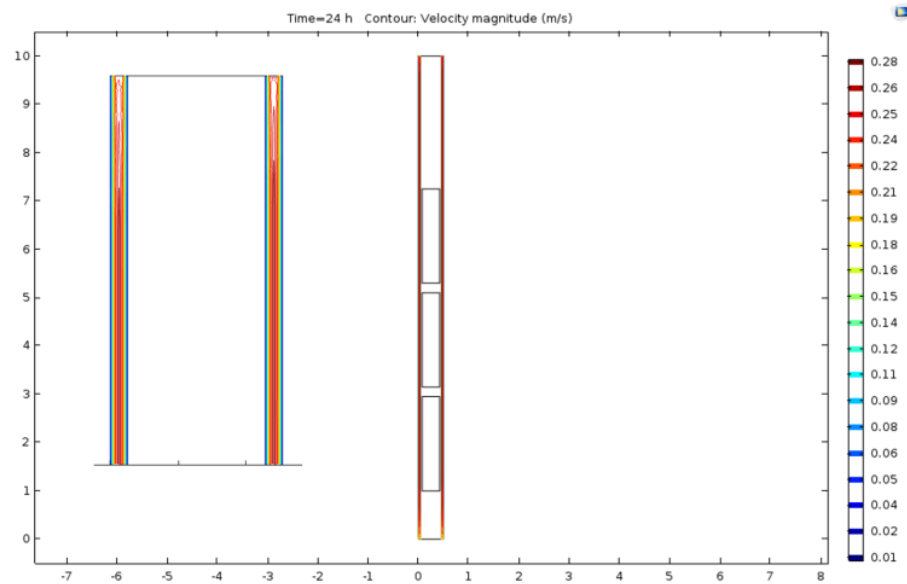


Figure 5.9: Velocity profile across ventilation pipe for 6 kW decay heat

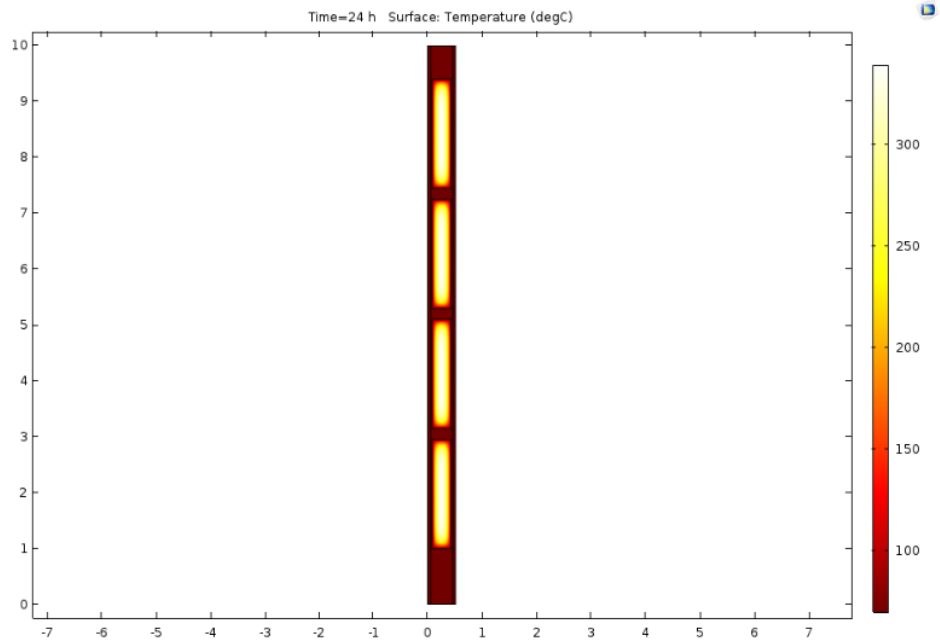


Figure 5.10: Temperature profile for 8 kW decay heat with passive cooling

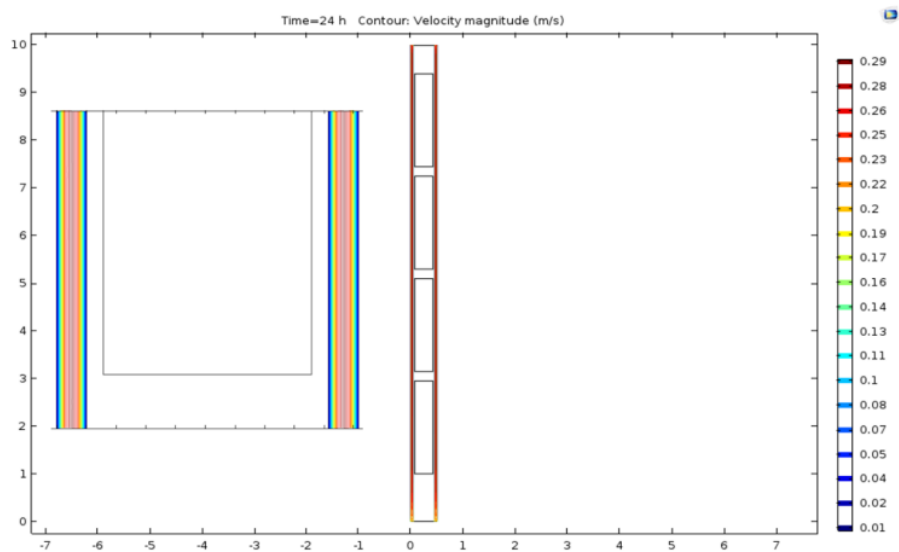


Figure 5.11: Velocity profile across ventilation pipe for 8 kW decay heat

6 SUMMARY AND CONCLUSIONS

Thermal analysis of passive cooling system has been implemented using a parametric analysis, and following parameters have been chosen. plenum height, duct dimensions, stack dimensions air gap between canister and thimble tube, annulus flow gap and number of canisters per locations. The canister centreline temperature is decreases with in the limit by the natural draft. The higher heat load condition increases the flow velocity thus increase in heat transfer coefficient and reduction of canister centreline temperature but it adds the higher air temperature due to reduction in the mass flow rate due to higher pressure drop in the annulus.

The experiment offers important data for a possible method of passive cooling to control heat generated by radioactive canisters in nuclear waste storage. Passive cooling systems offer a dependable and energy-efficient option while preserving the long-term security and integrity of the storage facility by utilizing natural heat transfer mechanisms.

5%

SIMILARITY INDEX

1%

INTERNET SOURCES

4%

PUBLICATIONS

1%

STUDENT PAPERS

PRIMARY SOURCES

- | | | |
|---|--|-----|
| 1 | P K WATTAL. "Indian programme on radioactive waste management", Sadhana, 2013
Publication | 1% |
| 2 | Green Energy and Technology, 2013.
Publication | 1% |
| 3 | Tzu-Chen Hung, Vijay K. Dhir, Bau-Shei Pei, Yen-Shu Chen, Fengjee P. Tsai. "The development of a three-dimensional transient CFD model for predicting cooling ability of spent fuel pools", Applied Thermal Engineering, 2013
Publication | <1% |
| 4 | coek.info
Internet Source | <1% |
| 5 | V. Verma, R. K. Singh, K. K. Vaze. "Thermal analysis of VWSB-IP1 at Tarapur", Kerntechnik, 2013
Publication | <1% |
| 6 | "Nuclear Fuel Cycle", Springer Science and Business Media LLC, 2023 | <1% |

7	Submitted to Manipal University Student Paper	<1 %
8	www.science.gov Internet Source	<1 %
9	Submitted to Mount St. Michael's College Student Paper	<1 %
10	Songkran Wiriyasart, Paisarn Naphon. "Numerical study on air ventilation in the workshop room with multiple heat sources", Case Studies in Thermal Engineering, 2019 Publication	<1 %
11	dokumen.pub Internet Source	<1 %
12	Submitted to UCSI University Student Paper	<1 %
13	www.ncbi.nlm.nih.gov Internet Source	<1 %
14	core.ac.uk Internet Source	<1 %
15	www.scribd.com Internet Source	<1 %
16	J. Bae, R. Bean, R. Abboud. "CFD analysis of a dry storage cask with advanced spent nuclear	<1 %

17

Raj, K.. "Radioactive waste management practices in India", Nuclear Engineering and Design, 200604

Publication

<1 %

18

Sami A. Al-Sanea, M.F. Zedan, M.B. Al-Harbi. "Heat transfer characteristics in air-conditioned rooms using mixing air-distribution system under mixed convection conditions", International Journal of Thermal Sciences, 2012

Publication

<1 %

Exclude quotes Off

Exclude matches Off

Exclude bibliography On

FINAL GRADE

GENERAL COMMENTS

/0

PAGE 1

PAGE 2

PAGE 3

PAGE 4

PAGE 5

PAGE 6

PAGE 7

PAGE 8

PAGE 9

PAGE 10

PAGE 11

PAGE 12

PAGE 13

PAGE 14

PAGE 15

PAGE 16

PAGE 17

PAGE 18

PAGE 19

PAGE 20

PAGE 21

PAGE 22

PAGE 23

PAGE 24

PAGE 25

PAGE 26

PAGE 27

PAGE 28

PAGE 29

PAGE 30

PAGE 31

PAGE 32

PAGE 33

PAGE 34

PAGE 35

PAGE 36

PAGE 37