ANRCP-1999-7
February 1999

Amarillo National Resource Center for Plutonium

A Higher Education Consortium of The Texas A&M University System, Texas Tech University, and The University of Texas System

A Feasibility Study for the Storage of Plutonium Pits in Non-Partitioned Warehouse Facilities

Darryl James, S. Parameswaren, and Suthaharan Nagendran
Department of Mechanical Engineering
Texas Tech University

Edited by

Angela L. Woods
Technical Editor

600 South Tyler • Suite 800 • Amarillo, TX 79101
(806) 376-5533 • Fax: (806) 376-5561
http://www.pu.org

This report was prepared with the support of the U.S. Department of Energy (DOE), Cooperative Agreement No. DE-FC04-95AL85832. However, any opinions, findings, conclusions, or recommendations expressed herein are those of the author(s) and do not necessarily reflect the views of DOE. This work was conducted through the Amarillo National Resource Center for Plutonium.

DISCLAIMER

ANRCP-1999-7

AMARILLO NATIONAL RESOURCE CENTER FOR PLUTONIUM/
A HIGHER EDUCATION CONSORTIUM

A Report on

A Feasibility Study for the Storage of Plutonium Pits in Non-Partitioned Warehouse Facilities

Darryl James, S. Parameswaran, Suthaharan Nagendran
Department of Mechanical Engineering
Texas Tech University
Lubbock, TX 79409

Submitted for publication to

ANRC Nuclear Program

February 1999

A Feasibility Study for the Storage of Plutonium Pits in Non-Partitioned Warehouse Facilities

Darryl James, S. Parameswaran, Suthaharan Nagendran
Department of Mechanical Engineering
Texas Tech University

Abstract

It is projected that up to 20,000 plutonium pits will be stored at Pantex for up to 50 years. The proposed storage system has to meet longevity, safety and cost requirements. Thermal, mechanical, chemical, nuclear criticality and safety performance characteristics of any proposed plutonium container design need to be formally analyzed. Plutonium generates thermal energy as it decays. The generated thermal energy may cause excessive rise of temperature. For safety and other considerations, it is important that the plutonium temperature remains relatively constant and no hot spots develop.

Plutonium containers should not be disassembled for routine monitoring and there are various reasons for the need to monitor the plutonium non-obtrusively. Therefore, accurate predictions of the temperature distribution within the storage container based upon external monitoring within the storage facility needs to be developed. A heat transfer analysis of the storage container is required. The heat transfer analysis, however, requires the knowledge of the temperature and velocity of the air circulating around the containers in order to determine the heat transferred to the air from the containers by convection. Therefore, a complete flow field analysis is required prior to performing the conduction analysis of each pit. The objective of this research is, therefore, to develop and validate a numerical model to predict the temperature distribution within the plutonium storage container as a function of the ambient air temperature within the warehouse.

TABLE OF CONTENTS

LIST OF FIGURES

1. FLOW MODELING

We are using a finite volume method to analyze the temperature distribution that will be generated in the storage facility. Three steps are necessary to generate results from the finite volume method. The first step is to create the geometry and the mesh. The second step is to solve for the temperature and velocity distribution within the computational domain. The final step is to input these results in the post processor to view the results graphically.

We used three commercially available software packages as our preprocessors for the generation of the geometry and mesh. The first one used was ICEM-CFD- Mulcad. This software package can handle up to 300 domains. The analysis requires 3000-4000 domains. Therefore this package could not be used. The next software package we used was ICEM-CFD- Hexa. This software does not allow the copying of the geometry of one cylinder to another and therefore each cylinder's geometry has to be created individually. Also, it is exceedingly difficult to check for the errors made while creating the geometry. It takes approximately three weeks for one storage option to be created. Since we will analyze several different storage options, this software package was found not suitable for our requirements.

Now, we are using a new software package, ICEM-CFD- Autohexa. This software allows the user to copy the geometry of one cylinder to another. It is also possible to create the geometry of one storage option within a day. Therefore we decided to use this software package for the creation of our geometry. The geometry of six different storage options was generated using this new software package.

Head3D is the solver used to compute the temperature and velocity distribution within a given computational domain. The output data from the preprocessor has to be transferred to the solver. At first, it was found that the output from Autohexa was not in a format acceptable to Head3D. Therefore, we worked with the manufacturer of the preprocessor to modify the interface to satisfy our requirements (note: some of the code developed by the ICEM CFD Engineering is proprietary and we are not able to view that source code). These modifications have taken several months. During this period, we received several modified versions of the interface. All of them were found to be unsatisfactory. At first, the solver could not read the data generated by the preprocessor. The later versions of the interface supplied by the manufacturer of the preprocessor gave output that could be read by Head3D, but Head3D did not run. Now, the latest version of the interface produces output that can be read and run by Head3D; however, the output generated by the solver was not consistent with the assigned boundary conditions. This is because of the errors while transferring the data from the preprocessors to the solver. i.e., the interface. We are still in the process of eliminating these errors.

With each interface received, two test geometries were used to check whether the interface worked properly. The first geometry we tested is shown in Figure 1.1. This geometry is an empty rectangular block with one inlet at the top and one outlet at the bottom. The output produced by the solver was found to be inconsistent with the assigned boundary conditions. We also tested another geometry, which is shown in Figure 1.2. In this case, a rectangular block has one inlet at the top and two outlets located at the bottom. Again, the output produced by the solver, found to be inconsistent with the assigned boundary conditions.

Six different numerical configurations for Room 121 of Building 12-116 have been assembled, three representing the actual configuration of supply and return air and three with two extra return air locations resulting in return air from each corner of the room. The first one is the configuration given by the clients. Frontal and plan views of this geometry are shown in Figures 2.1 and 2.2. The model has eight inlets at the top and two outlets located near the floor at two diagonally opposite corners. The plutonium pits were stored in stacks of six cylinders. There are two layers of these stacks in the storage facility. There are 504 cylinders kept in this storage option.

In the second geometry, shown in Figures 3.1 and 3.2, there are four outlets located in the corners of the storage facility, unlike the first geometry which had only two outlets. Having four outlets will improve the flow patterns within the storage facility. A Stage Right rack configuration is modeled in both Figures 2.1 and 3.1.

The remaining configurations represent "what-if" scenarios that can be generated easily to test the effect of rack configuration on fluid flow and fluid temperature. The plan views of the remaining configurations are the same as either Figure 2.2, if two return air locations are present, or Figure 3.2 if four return air locations are present. The frontal view for each "what if" configuration is shown in Figures 4.1 (two return air locations), 5.1 (four return air locations), 6.1 (two return air locations) and 7.1 (four return air locations). The inlets and the outlets are similar to that of the first geometry, but the arrangement of the cylinders is different.

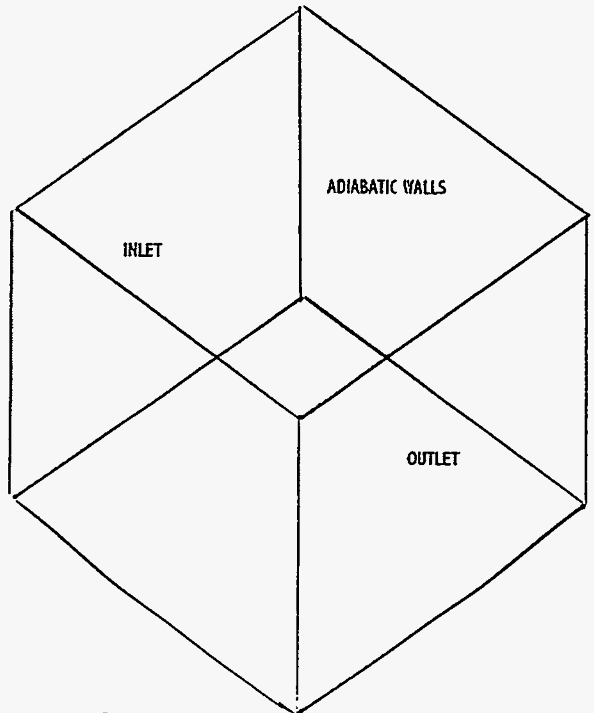

Figure 1.1 Isometric view of test Geometry 1

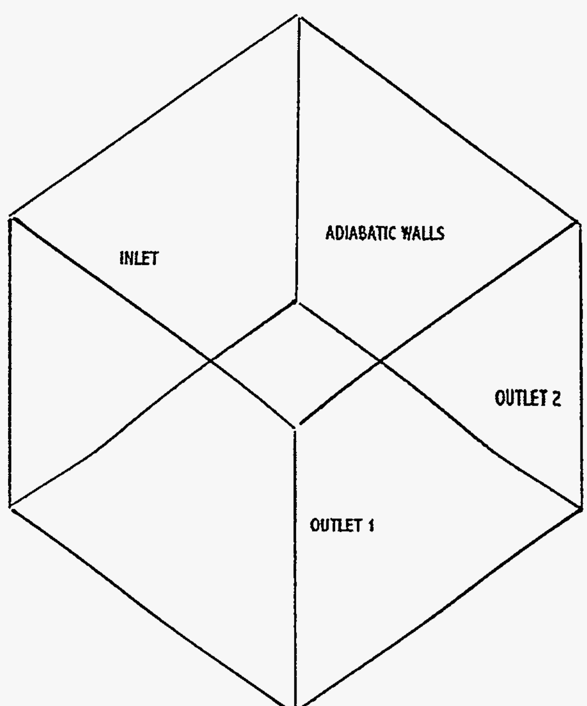

Figure 1.2 Isometric view of Test geometry 2

Figure 2.1 Front view of Geometry 1

Figure 2 2 Plan view of Geometry 1

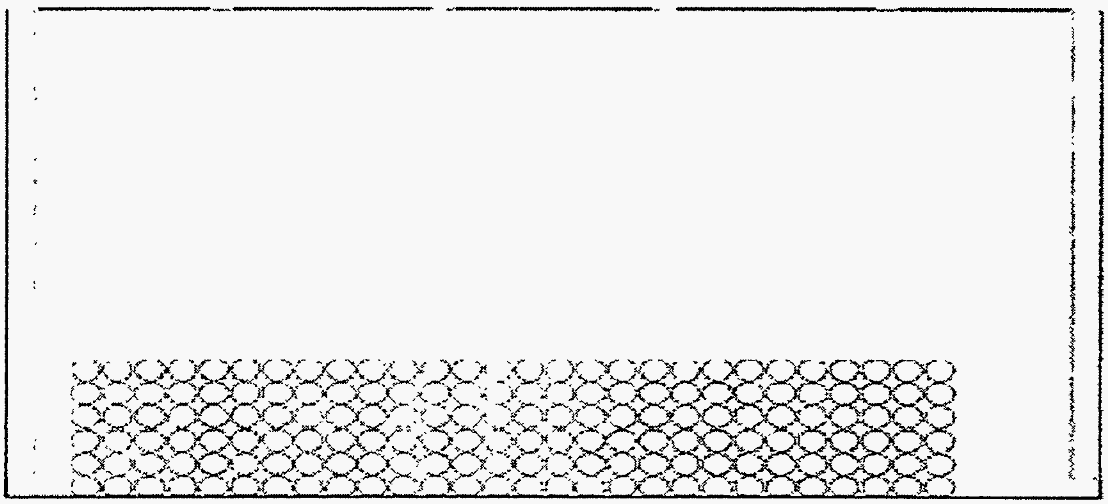

Figure 3.1 Front view of Geometry 2

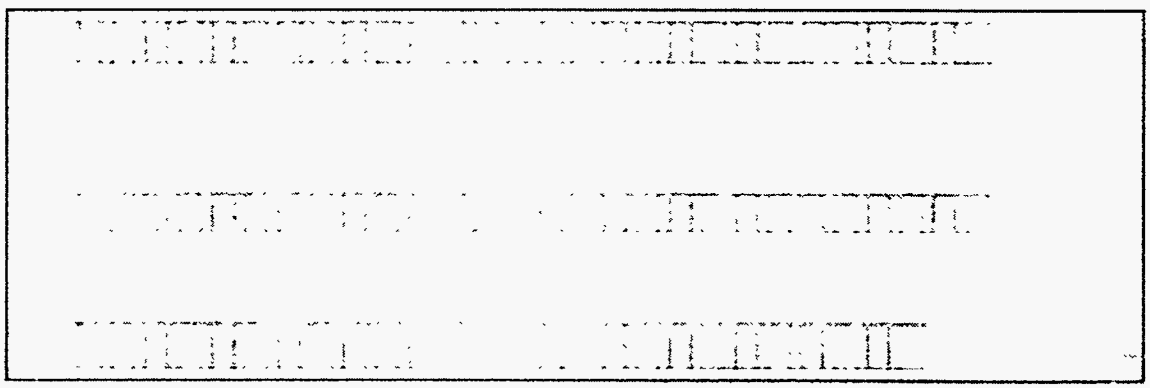

Figure 3.2 Plan view of Geometry 2

Figure 4.1 Front view of Geometry 3

Figure 5.1 Front view of Geometry 4

Figure 6.1 Front view of Geometry 5

Figure 7.1 Front view of Geometry 6

2. HEAT TRANSFER AND FLUID FLOW COUPLING

As stated previously, the objective of this research is to develop and validate a numerical model to predict the temperature distribution within the plutonium storage container as a function of the ambient air temperature within the warehouse. A necessary step toward this goal is to combine the fluid flow analysis with a conduction heat transfer analysis. In order to perform this analysis effectively, we must be able to simultaneously mesh the solid structures and the fluid for each computational domain. It is therefore required that the new interface that we are developing with ICEM-CFD Engineering be able to mesh everything within the computational domain and that we know the common boundaries shared by both fluid and solid structures.

A simple test geometry (computational domain) consisting of a cylinder contained within a rectangular block is being considered for this purpose. The rectangular block and the cylinder were divided into 800 cells. A code is being written to identify the cells at the boundary between the cylinder and the fluid. Code is also being written to identify cells on the cylinder and the rectangular block that are adjacent to each other (share the same boundary). This code is being developed and is being tested with every new interface that we receive. The geometry and the mesh for this configuration are shown in Figures 8.1-8.3.

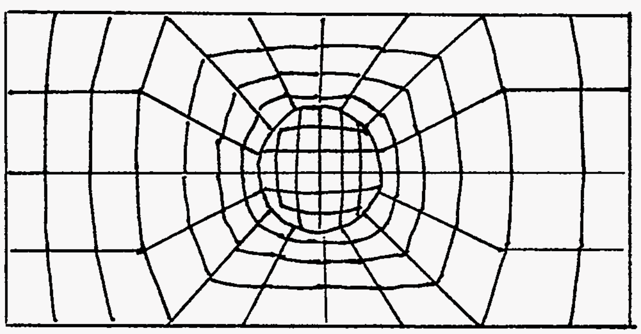

Figure 8 1 Front view of the conduction model

Figure 8.2 Isomatric view of the conduction model with mesh

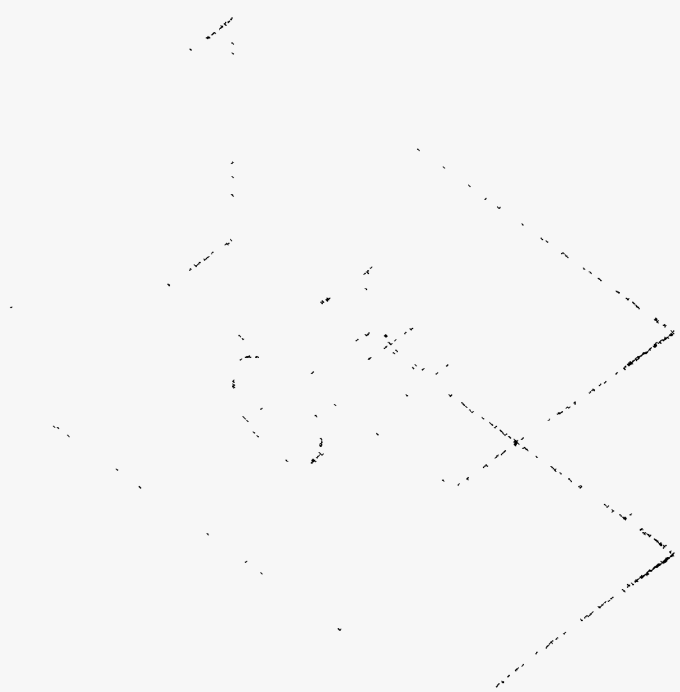

Figure 8.3 Isomatric view of the conduction model with mesh in cylinder

REFERENCES

There were no references cited for this report.