U.P.B. Sci. Bull., Series …, Vol. …, No. …, 2009 ISSN 1454-23xx

EVALUATION OF HEAT TRANSFER IN A DRY STORAGE MODULE OF CANDU FUEL EMPLOING ANSYS-CFX

Gheorghe-Ghiocel Simionescu[1], Radu-Mihail Bilegan[2]

The paper is representing a first step in the validation of the method to be applied in the elaboration of design calculations and/or in the thermal checking of the increased storage CANDU modules by using ANSYS-CFX.

1. Introduction

The Mechanical Analysis Department within Center of Technology and Engineering for Nuclear Projects (CITON) has developed methods and purchased computer programs dedicated to the elaboration of the thermo-hydraulic analyses both for thermal design of the equipment or piping systems and thermal stress determination.

Within such concerns the department has also been involved in the conjugate hydraulic and thermal analyses for checking the CANSTOR 200 module for the dry storage of spent fuel in the Cernavoda NPP Site. This paper is such an analysis in a CANSTOR 200 module to identify the characteristic feature of the ANSYS-CFX program application and temperature determination inside the module. Also, the calculated temperatures compliance with experimental values and satisfaction of the allowable limits [1], [2] were verified.

The paper is representing a first step in the validation of the method to be applied in the elaboration of design calculations and/or in the thermal checking of the increased storage capacity modules which, in fact, is a worldwide current concern [3], [4].

The analyses presented in this paper have been conducted by ANSYS WORKBENCH 1.0 along with its sub-sections – ANSYS Design Modeler and ANSYS-CFX [5].

ANSYS-CFX is able to perform 3D flow and conjugate heat transfer analyses for transient and steady-state problems, being from this point of view better than other programs used in solving similar problems [3], [4].

---

[1] *PhD., Stress Analysis Department, , Center of Technology and Engineering for Nuclear Projects , Bucharest, Romania*

[2] *Senior Engineer, Stress Analysis Department Manager, Center of Technology and Engineering for Nuclear Projects , Bucharest, Romania*

2. Reference Requirements for Analysis.

2.1. Module Description

A cross-section view of the module subjected to the analysis may be seen in fig.3.1.-1, 2, 3 below.

The module is shaped as a parallelepiped concrete box of 21.64 m X 8.14 m X 7.45 m in size. The concrete wall thickness is minimum 965 mm on the sides and ends, 1067 mm on ceiling and 157 mm on the floor.

Inside there are 20 storage cylinders anchored to the ceiling and equally arranged on 2 rows. Each cylinder may accommodate 10 storage baskets and each storage basket may include 60 spent fuel bundles so that the capacity of a storage unit is 12 000 bundles/module.

In the lower part of each module side walls there are 5 ventilation openings for the cool air incoming into the module and 6 exhaust openings on top for to exhaust the hot air from the module.

2.2. Assumptions

2.2.1. Computation Model

In this paper, the heat transfer inside the module structure from the storage cylinders up to the outer surface of the module is analyzed , also considering the heat transfer on the module outside.

The followings have been considered:

- Steady state;
- Conduction with internal sources inside the cylinders;
- Natural convection from the storage cylinders and concrete wall inside surfaces towards the air inside the module;
- Conduction from the storage cylinders to the concrete ceiling above cylinders;
- Radiations from the storage cylinders to the concrete wall internal surfaces (inside air considered transparent to radiations);
- Conduction inside the concrete walls of the module;
- Natural convection and radiation from the concrete wall outer surfaces to the environment.

The heat transfer inside the storage cylinders is being analyzed in case of an homogeneous structure with heat sources uniformly distributed in the volume of each cylinder. So, the temperature distribution obtained inside the storage cylinders is not the actual one but the model is offering the boundary thermal and flow conditions on the cylinder- fluid separation surfaces according to the physical reality so that the analysis on the flow and heat transfer starting from the cylinders towards the outside of the module is in compliance with the physics laws.

Under such circumstances the analysis on the heat transfer inside the storage cylinders represents another problem that is to be analyzed either by a dedicated model or on an extended model of this model.

2.2.2. Fuel Burnup Degree and Cooling Period.

The fuel is a standard natural Uranium CANDU6 bundle that has benefited from a cooling-down period of 6 years.

Under such circumstances the residual thermal power considered in this paper is 3648 W/cylinder, according to some references in the literature [1], [2].

2.2.3. Cooling-Down Conditions.

For normal operation conditions the analysis postulated a temperature of the ambient air of 40 degrees Celsius and the air inlets and outlets were considered to be 100% open (unplugged) but provided with protection grids according to the physical conditions.

2.3. Imposed Limits

As per ref. [1], [2], [3], [4] the following maximum temperature values were considered:

- 180 degrees Celsius for the fuel element temperature;
- 100 degrees Celsius for the local temperature of the concrete;
- 65 degrees Celsius for the volumetric temperature of the concrete.

3. ANSYS WORKBENCH Models

3.1. Geometric Model

The geometric model is given in section Design Modeler of the computer program (see Fig 3.1-1).

Due to the complexity of CANSTOR 200 module and its very big sizes, in order to reduce the calculation volume, the model was limited to a central section consisting of 4 storage cylinders as shown in fig. 3.1-1, 3.1-2 and 3.1-3.

In ANSYS language the geometric model used in the analyses is including 6 bodies, namely:

- 4 bodies representing the 4 cylinders contained in the model (see fig.3.1-1);
- 1 body representing the air area inside the model (see fig. 3.1-2);
- 1 body representing the concrete walls of the model (see fig.3.1-3).

The meshing was developed in 2 steps: first, was a surface meshing followed by a volume meshing, with 5 “Inflated Layers” on the fluid-concrete walls interfaces.

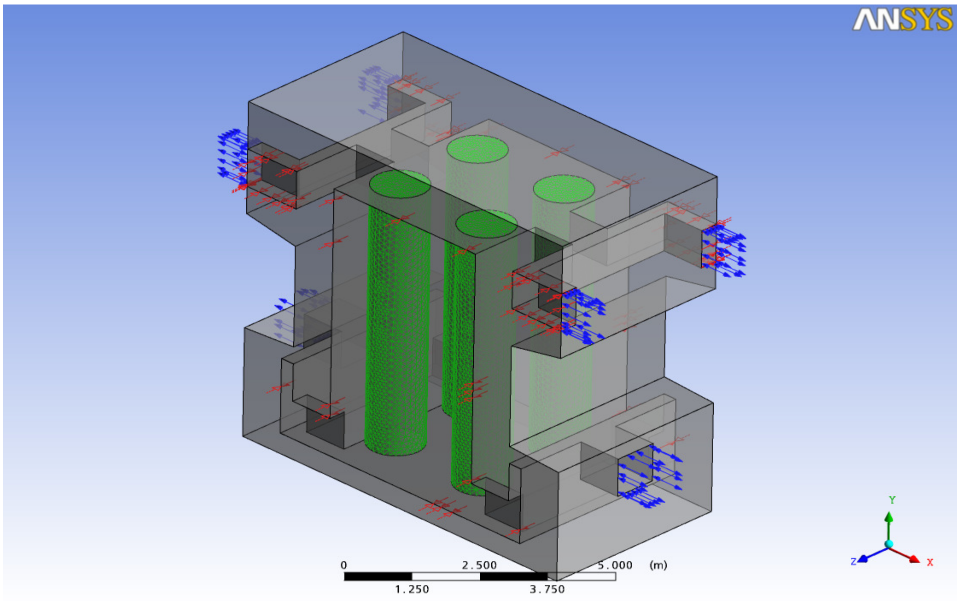

Fig. 3.1-1 – Geometric Model

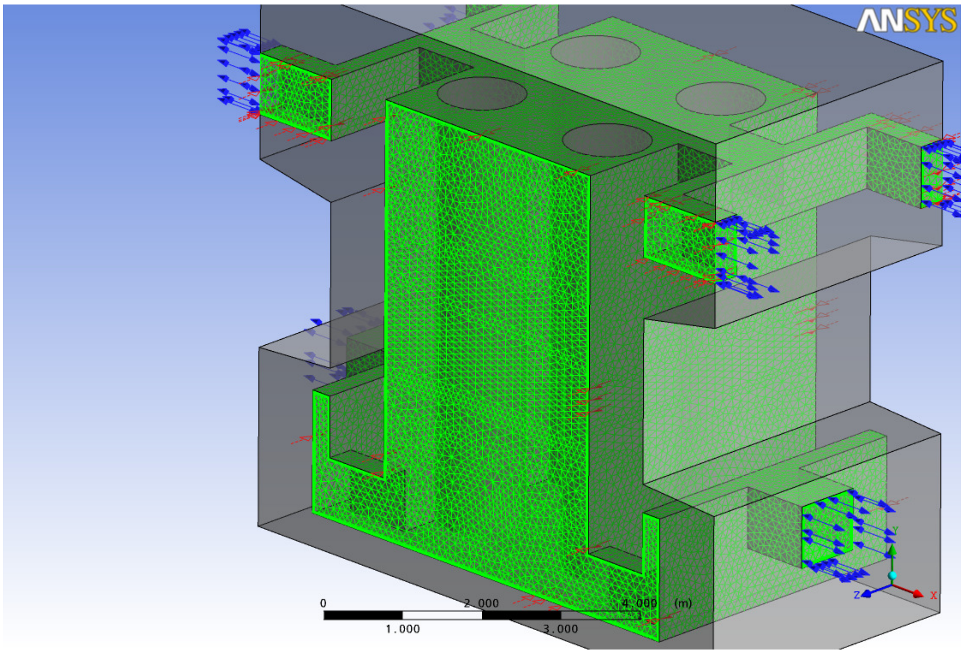

Fig 3.1-2 – Air zone inside the model

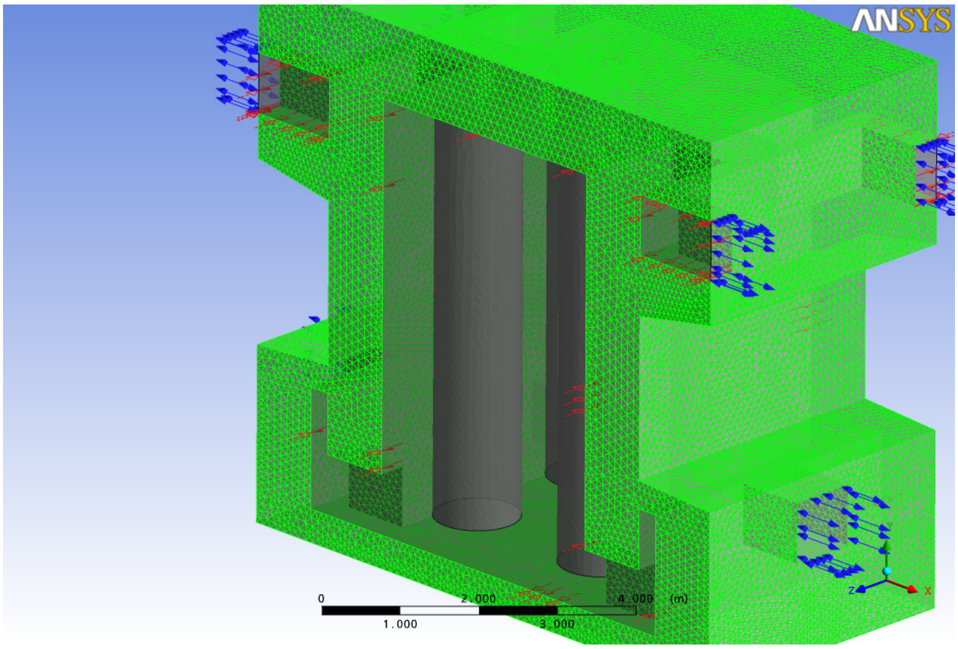

Fig. 3.1-3 Concrete walls of the model

3.2. Physical Model

The physical model was made by ANSYS-CFX pre-processor. This allows the conversion of geometric bodies made by the Design Modeler in domains and sub-domains (fluids, solids or porous) with specific material properties to which motion equations and thermo-dynamic equations are associated, where the case. By means of the pre-processor, the user may also define the type of the physical problem (transient or steady state) and function of the established type, the initial conditions and the interface and boundary conditions for studied process.

3.3. Solving the Physical Model.

Solving the physical model implies the initialization of the parameters and the development of some iterations in order to integrate the solution in the imposed convergence criteria.

In this paper, for the initialization the Automatic option was selected for all the defining parameters of the process. The maximum number of iterations was established at 250 and the selected convergence criterion was Residue RMS = 1e-05.

4. Results

With ANSYS-CFX post-processor some processings were done:

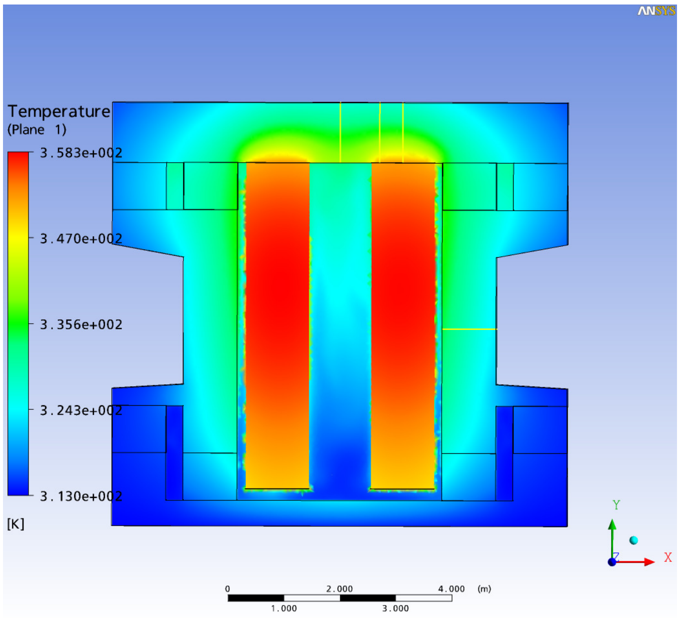

Fig. 4-1

Temperature Map in a Transversal-Vertical Section through the Module

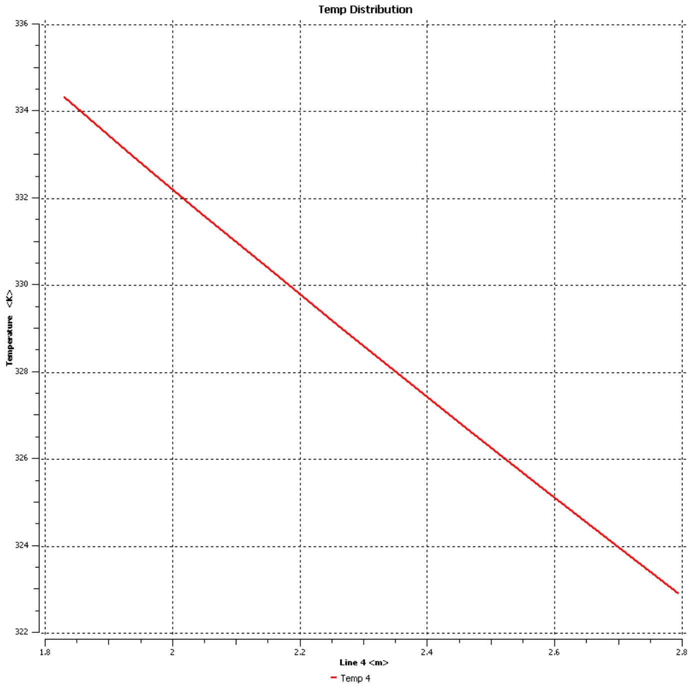

Fig 4-2

Temperature distributions in the lateral wall along the wall thickness, in an central area

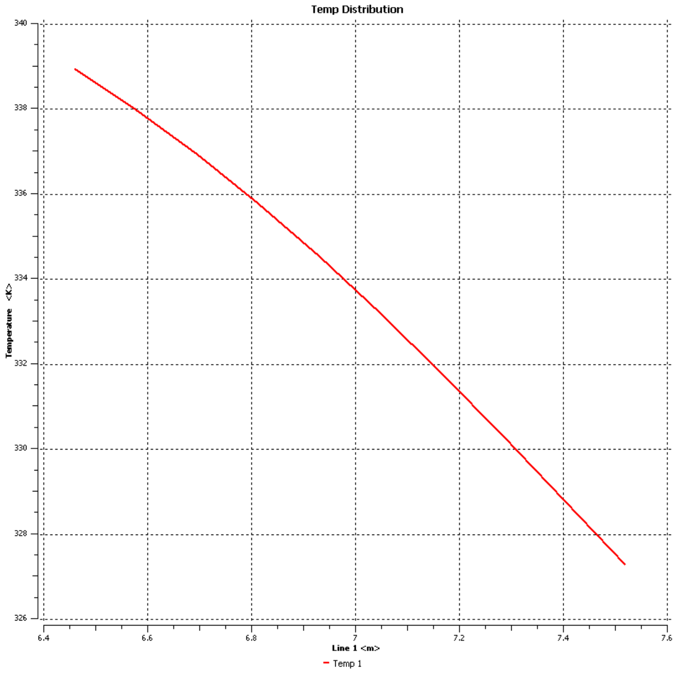

Fig. 4-3

Temperature distribution in the ceiling along the wall thickness in an area between the storage cylinders.

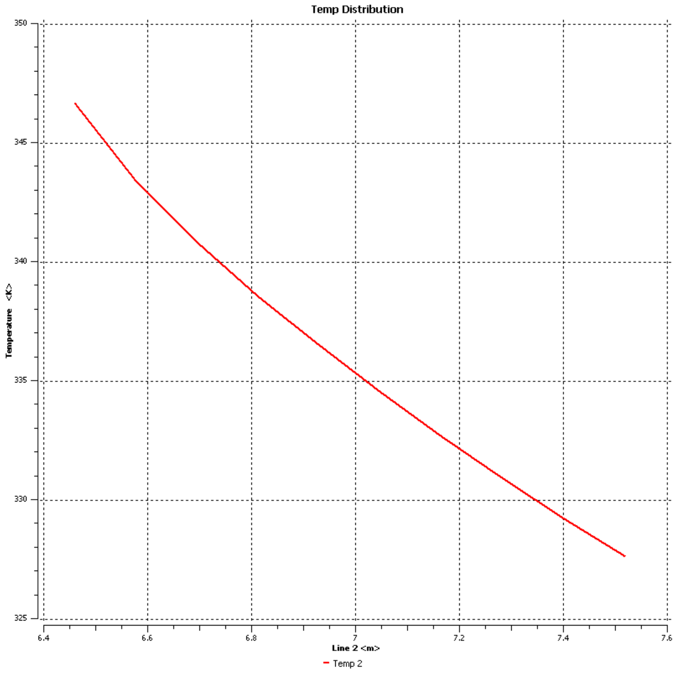

Fig 4-4

Temperature distribution in the ceiling along the wall thickness in front of one of the storage cylinders.

It was found that :

- the ceiling temperatures and the lateral wall temperatures are complying with the experimental values [1], [2] with a maximum deviation of +3 Celsius degrees and +1 Celsius degree, respectively;
- concrete wall temperatures are in the range of :

  334.5 K-323 K, for lateral walls in a central area;
  339 K-327.5 K, for a ceiling between the cylinders;
  346.5 K-327.5 K, for a ceiling in front of one of the cylinders;
- the air circulating through the module is getting heated by 22 Celsius degrees:
- the temperature at the inner surfaces of the lateral walls is 3.5 Celsius degrees less than the limit of 65 Celsius degrees imposed for the concrete volume temperature;
- the temperature at the inner surfaces of the ceiling between cylinders is 34 Celsius degrees less than the limit of 100 Celsius degrees imposed for the concrete local temperature;
- the temperature at the inner surfaces of the ceiling above cylinders is 26.5 Celsius degrees less than the limit of 100 Celsius degrees.

5. Conclusions

The paper is a presentation of the heat transfer analysis in the CANSTOR 200 storage module in normal operation conditions.

For the analysis ANSYS WORKBENCH program along with its sub-sections- ANSYS Design Modeler and ANSYS CFX were used.

The findings show that the calculated temperatures have a good coincidence with the experimentally determined temperatures.

In conclusion, this paper shows a method for the proper thermal analysis to be applied in the design calculations and/or thermal checking of the increased storage CANDU modules.

REFERENCES

[1] *AECL-66-62960-220-001 / 0* – Étude du Comportament Thermique du Module CANSTOR#2
[2] *AECL-66-62500-220-002 / 0* – Étude du Comportament Thermique du Module CANSTOR#2
[3] *G.Sabourin et al.* – Simulation of transient heat transfer in MACSTOR/KN-400 Module Storing Irradiated CANDU Fuel
[4] *R.Beaudoin, R.Shill et al.* – Evaluation of Safety Margins during DRY STORAGE of CANDU Fuel in MACSTOR/KN-400 Module.