14th International Symposium on the Packaging and Transportation of Radioactive Materials (PATRAM 2004), Berlin, Germany, September 20-24, 2004

Paper # 158

Analysis of removal of residual decay heat from interim storage facilities by means of the CFD program FLUENT

Werner Stratmann, Peter Hages

STEAG encotec GmbH, Essen, Germany

1 Introduction

Within the scope of nuclear licensing procedures of on-site interim storage facilities for dual purpose casks it is necessary, among other things, to provide proof of sufficient removal of the residual decay heat emitted by the casks. The results of the analyses performed for this purpose define e.g. the boundary conditions for further thermal analyses regarding the permissible cask component temperatures or the maximum permissible temperatures of the fuel cladding tubes of the fuel elements stored in the casks. Up to now, for the centralized interim storage facilities in Germany such analyses were performed on the basis of experimental investigations using scaled-down storage geometries [1]. In the engineering phase of the Lingen on-site interim storage facility, proof was furnished for the first time using the CFD (computational fluid dynamics) program FLUENT. The program FLUENT [2] is an internationally recognized and comprehensively verified program for the calculation of flow and heat transport processes.

Starting from a brief discussion of modeling and the different boundary conditions of the computation, this contribution presents various results regarding the temperatures of air, cask surfaces and storage facility components, the mass flows through the storage facility and the heat transfer at the cask surface. The interface point to the cask-specific analyses is defined to be the cask surface.

2 The storage building

The storage building is divided up into a receiving and a storage area. The storage area (**Fig.1**) is broken down in 10 bays, each one 6.4 m wide, and designed to accommodate a total of 100 casks. In longitudinal direction, the cask positions are arranged at a (center-center) distance of 3.2 m, in transverse direction at a distance of 3.0 m. In the storage area, the casks are lined up in 20 rows with five casks each. The average thermal output of the casks is approx. 375 kW per 10 casks (2 rows). Since a conservative approach had been chosen for the computations, the decay of the thermal power was not taken into account.

Air inlet openings have been arranged in one longitudinal wall of the storage hall for supply of cold inlet air, and exhaust openings in hall roof near the opposed wall of the hall for removal of the heated up air (**Fig. 2**). The air inlet and outlet openings can be closed by louvers.

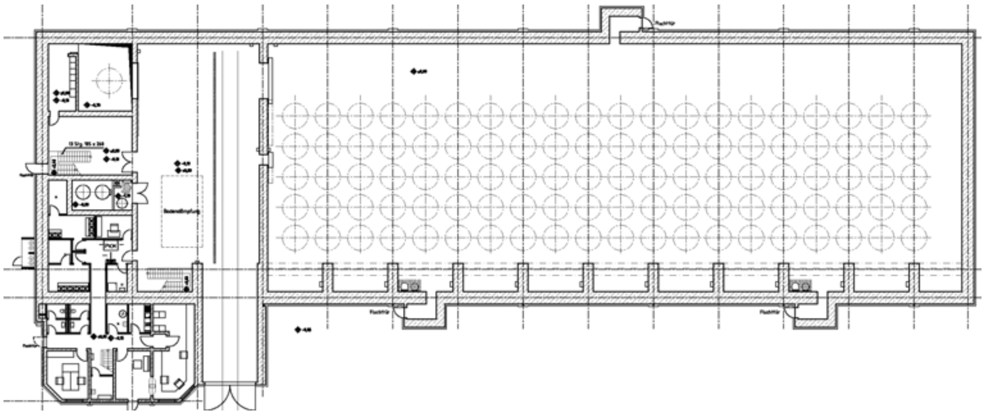

Fig. 1. Ground plan of an interim storage facility for 100 casks

3 Software used

The software package "FLUENT" was used to solve the problem. “FLUENT” is a system for numerical flow simulation, which solves the following integral conservation equations:

- mass conservation,
- momentum conservation,
- energy conservation,
- radiation transport equation.

The equations are discretized according to the finite-volumes method using a computational grid adjusted to the bodies. The equation system is solved by special, sophisticated algorithms. A PC version of the program was installed on a double-processor system with 2 GByte of RAM. On such system, a computation run typically takes one to two days.

For description of the flow processes within the storage area, a k-ε turbulence model was used. The interchange of radiation among the casks and between the casks and the enveloping hall structure was computed by means of a discrete ordinates model.

4 Description of the computational model

Geometry

Due to the size of the object to be simulated, simplifying assumptions had to be made for modeling. Based on the periodical geometry of the hall and the marginal areas that can be neglected in the conservative approach, only one segment of the total hall volume was modeled.

Another assumption referred to the fins on the cask surface, the geometry of which is not explicitly covered by the computational model. The increased heat transfer and the impact on the flow caused by the fins were numerically allowed for by choosing a suitable surface roughness.

Apart from the simplifying assumptions described above, the geometric model was not simplified further. **Fig. 2** shows an overall view of the model segment.

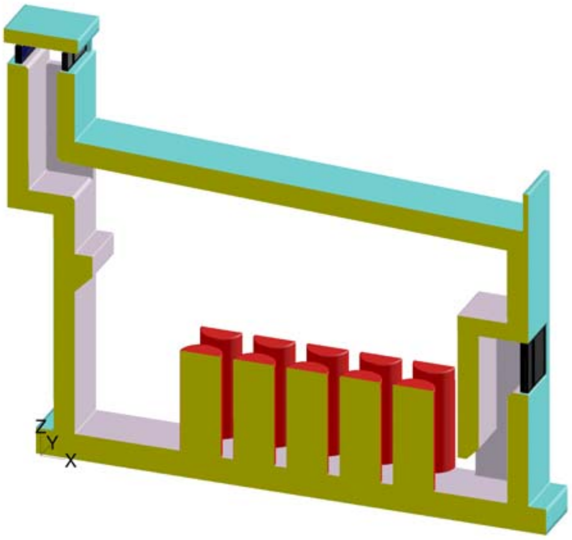

Fig. 2. Overall view of the model hall segment

Fluid and solid body properties

In the computation all fluid properties are treated as a function of temperature. Taking into account the pressure level of ≈ 10^5 Pa used as a basis for the computation, air may be considered to be an ideal gas.

As storage casks, initially a cask with internal moderator was investigated. For the different zones in the cask, the effective heat conductivity of the respective material was used. The heat output of 37.5 kW emitted from one cask was distributed in the form of volume-type sources to the four discretized cylinders within the cask.

The cask surface was divided into three areas, i.e. top-end surface, bottom-end surface and shell surface. Top-end and bottom-end zones were treated as hydraulically smooth surfaces. For the shell surface, a surface roughness was input to approximate the fluid dynamic and thermal properties of the finned cask surface.

The degree of detailing of the cask model was chosen such that the surface temperature and the heat transfer at the cask surface were accurately reflected. Detailed modeling of the cask interior was dispensed with. The computation of component temperatures and fuel rod temperatures within the cask is the responsibility of the cask manufacturers.

5 Results

Three-dimensional flow

Fig. 3 shows the air flow paths, with the colors indicating the air temperature. The cold air flow enters the storage area through the air inlet openings; passing along the casks, it is heated up and accelerated upward before it leaves the storage building through the exhaust openings.

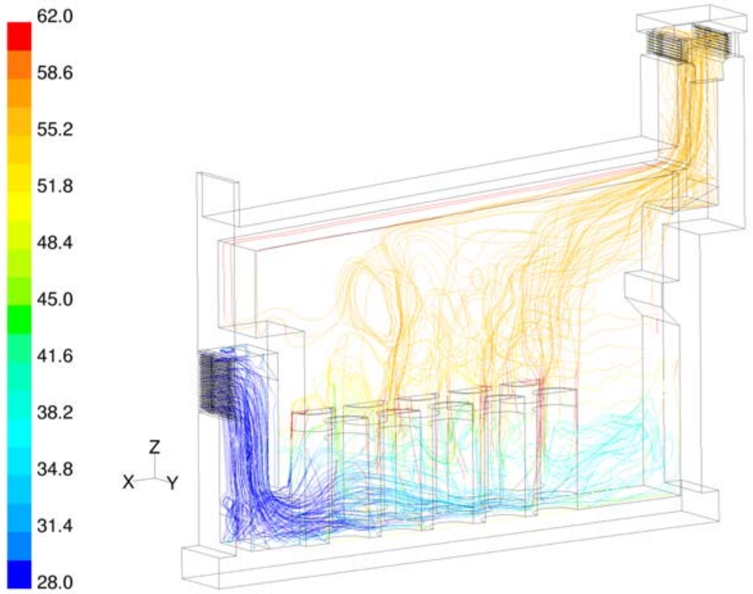

Fig. 3. Three-dimensional flow lines, colors indicating temperature

Air temperature

Fig. 4 shows the temperature distribution in a cross section of the hall, which is characterized by distinctive temperature stratification. Average temperatures near the floor of the hall range from 28 °C near the inlet air duct to a maximum of approx. 40 °C at the opposing side of the hall. Also the casks at the rear of the hall are sufficiently supplied with cold air. In the area above the casks, a temperature of approx. 56 °C is reached, which approximately corresponds to the air outlet temperature.

Concrete temperatures
The temperature distribution in the civil structure is also evident from **Fig. 5**. In the area of the air inlet duct, a distinctive temperature profile emerges in the partition walls. Below the casks, the temperature rises to values of approx. 85 °C, caused by the heat transfer from the cask to the floor. The rear side of the hall is heated up by the thermal radiation from the casks. In the lower area of the wall, the heat can be transferred by natural convection to the relatively cold air and temperatures in this are will thus not exceed 64 °C. In the upper area, which is in contact with the relatively hot air, the convection transfer of heat from the wall to the air is more limited, resulting in a maximum temperature of approx. 74 °C in this area.

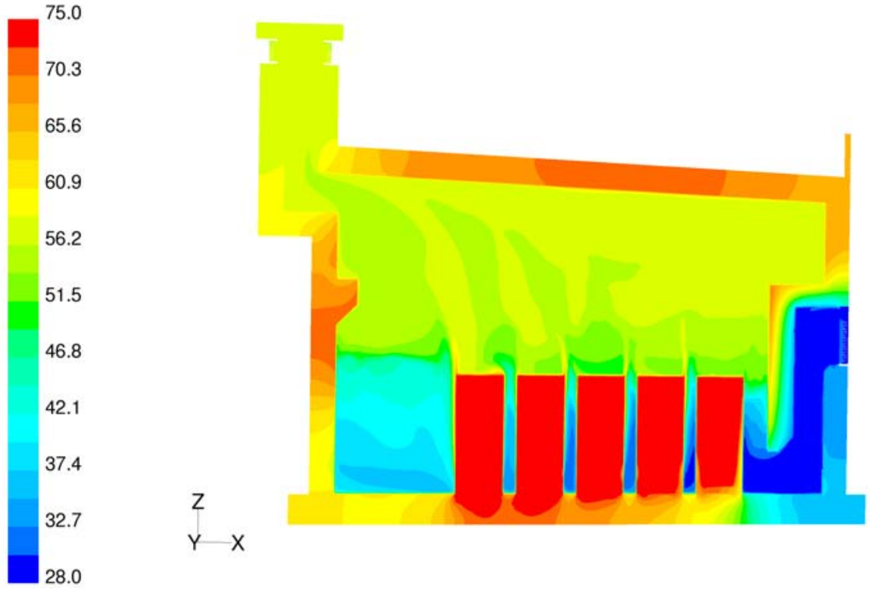

Fig. 4. Temperature distribution in air and in the civil structure (values ≤ 75°C)

Even more distinctive is the temperature rise of the concrete in the roof area. In the area of the left hand side of the roof, the convection cooling effect is quite poor, which results in a maximum temperature of approx. 74 °C there.

Cask surface temperature
For the shell surface of the casks, the computation – relating to all casks – results in a maximum average surface temperature of approx. 87 °C. Limited local peak temperatures will not exceed approx. 97 °C.

Heat transfer coefficient
The heat transfer coefficient α at the cask surface was calculated for a reference temperature of 28 °C. The heat transfer coefficient α includes the heat transfer paths radiation and convection. Averaged over all casks, the following values result for the heat transfer coefficient in the three surface zones:

Cask bottom zone:	$\alpha = 7.3\ \mathrm{W/m^2K}$,
Finned shell zone:	$\alpha = 20.4\ \mathrm{W/m^2K}$,
Cask top zone:	$\alpha = 3.9\ \mathrm{W/m^2K}$.

6 Summary

The CFD simulation of the flow and heat transport processes produces detailed results with regard to the resulting air and concrete temperatures, the flow field in the storage hall, the temperatures of cask surfaces and the heat transfer at the cask surfaces.

Especially in case of a mutual impact on the structural design due to high shielding requirements, the CFD simulation provides the possibility to optimize heat removal from the storage area by means of structural design measures.

In a further step, the CFD simulation might be used to simultaneously provide proof of heat removal from the storage facility and proof of heat removal from the cask – which today are still two separate proofs – by using a detailed cask model in the CFD simulation of the storage facility. This would provide a one-step solution to the entire heat removal problem and might thus allow dispensing with unnecessary conservative assumptions concerning the cask proofs.

Bibliography

[1] W. v. Heesen et al. , “Heat Transfer from Transport Cask Storage Facilities for Spent Fuel Elements,“ NUCLEAR TECHNOLOGY VOL.62 JULY 1983

[2] FLUENT 5 User’s Guide Fluent Incorporated, Lebanon, NH 03766
July 1998