Validation of CFD-Methods to Predict Heat Transfer and Temperatures during the Transport and Storage of Casks under a Cover

André Leber
WTI Wissenschaftlich-Technische-Ingenieurberatung GmbH, Germany

Wilhelm Graf
GNS Gesellschaft für Nuklear-Service mbH, Germany

Roland Hüggenberg
GNB Gesellschaft für Nuklear-Behälter mbH, Germany

1. Introduction

With respect to the transport of casks for radioactive material, the proof of the safe heat removal can be accomplished by validated calculation methods. The boundary conditions for thermal tests for type B packages are specified in the ADR based on the regulations defined by the International Atomic Energy Agency.

The varying boundary conditions under transport or storage conditions are based on the varying thermal conditions true for different cask types.

In most cases the cask will be transported in lying position under a cover (e.g. canopy or tarpaulin) and stored in standing position in an array with other casks. The main heat transport mechanisms are natural convection and thermal radiation. The cover or the storage building are furnished with vents that create an air flow, which will improve the natural convection. Depending on the thermal boundary conditions, the cask design and the heat power, about 50 - 95 % of the heat power will be removed from the finned cask surface by natural convection. Consequently the convection by air flow is the main heat transport mechanism.

The air flow can be approximated with analytical methods by solving the integral heat and flow balances for the domain. In a stationary state the overpressure due the buoyancy and the pressure loss in the flow resistances are equal. Based on the air flow, the relevant temperatures of the cask can be calculated in an iterative process.

Due to the fast development of numerical calculation methods and computer hardware, the use of Computational-Fluid-Dynamics(CFD) calculations plays an important role. CFD-calculations are based on solving the equations of conservation (Navier-Stokes equations) using a finite element mesh or a finite volume mesh of the model. For a finned cask lying under a cover, where the main contributing element for heat removal is natural convection in combination with the thermal radiation, a CFD-calculation can be the most appropriate method.

Common CFD-Codes are FLUENT or CFX using finite-volume solvers or ANSYS-FLOTRAN using finite-element solvers. The correct functioning of these codes is globally tested over a bread rage of industrial flow problems. The weakly forced natural convection coupled with convective and radiative heat transfer at a finned surface is a very special use of CFD, so that the numerical methods by using CFD have to be validated in detail for these applications.

2. Heat Transport Mechanisms

The scheme of the main heat transport mechanisms is shown in Figure 1. The heat power is transported through the cask body to the finned cask surface by heat conduction. The higher temperature level of the cask surface leads to buoyancy effects, which occur in any variable-density flow in a gravity field. Cold inlet air flowing over the finned cask surface, is heated up and rises to the outlet vents. Another part of the heat power will be directly transported by radiation from the cask surface to the inner surface of the cover.

The total heat power - meaning the decay heat of the casks inventory and the solar absorption at the surface of the cover - is removed with the outlet air flow and by convection and radiation from the outer surface of the cover.

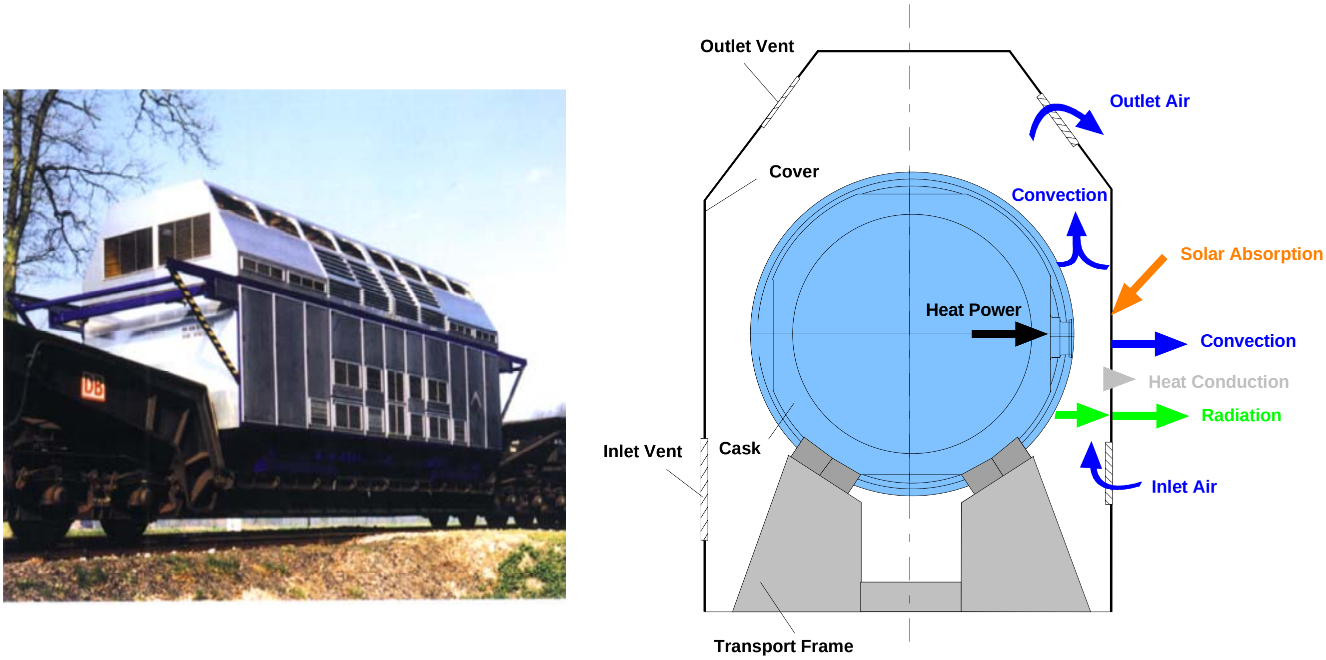

Figure 1: Heat Transport Mechanisms during a Cask Transport

3. Basic equations of buoyancy and conversion to CFD

The following equations are deduced from the fundamental balances of mass, momentum and energy (Navier-Stokes equations). They are formulated for a thin-shear-layer of a Newtonian Fluid in a two dimensional flow. With the x-axis chosen vertically upward, the momentum equation for a laminar flow becomes:

$$\rho\left(u\frac{\delta u}{\delta x}+v\frac{\delta u}{\delta y}\right)=-\frac{dp}{dx}+\frac{\delta}{\delta y}\left(\mu\frac{\delta u}{\delta y}\right)-g\,(\rho-\rho a)$$

with ρ: density
u, v: velocities
p: pressure excluding the hydrostatic contribution
μ: viscosity
g: acceleration of gravity
ρa: ambient density

If the density differences are very small, the Boussinesq approximation can be used in combination with the thermal volumetric expansion β. This approximation is available in most CFD-programs.

$$\rho - \rho_a = -\rho_a \beta (T - T_a) \quad \text{with} \quad T: \ \text{temperature}$$

$$T_a: \ \text{ambient temperature}$$

$$\text{and} \quad \beta = -\frac{1}{\rho}\left(\frac{\delta\rho}{\delta T}\right)_p \qquad (\beta = 1/T \ \text{for an ideal gas})$$

Under consideration of the approximation above and an additional turbulence term, characterised by the dyadic product of the velocity change u´·v´, the equation of momentum for a turbulent flow becomes:

$$\rho_a\left(u\frac{\delta u}{\delta x} + v\frac{\delta u}{\delta y}\right) = -\frac{dp}{dx} + \mu_a\left(\frac{\delta^2 u}{\delta y^2}\right) - \rho_a\, g\, \beta (T - T_a) + \rho_a \frac{\delta}{\delta y}(\overline{-u'v'})$$

The continuity equation and the energy equation are:

$$\frac{\delta u}{\delta x} + \frac{\delta u}{\delta y} = 0$$

$$u\frac{\delta T}{\delta x} + v\frac{\delta T}{\delta y} = \frac{\lambda_a}{\rho_a c_a}\left(\frac{\delta^2 T}{\delta y^2}\right) + \frac{\delta}{\delta y}(\overline{-T'v'})$$

with T': temperature change

λ_a: conductivity

c_a: specific heat

These basic equations of conservation can be solved numerically with common CFD-Codes for a discrete model using a turbulence model for the turbulence and viscosity terms.

4. Modelling of turbulence, radiation and surface enlargement of finned surfaces

For the CFD simulation one- or two-equation turbulence models with a wall function to simulate the boundary layer are employed. The commonly used turbulence models are based on the standard *k*-ε-model, with *k* as the turbulent kinetic energy and ε as the dissipation rate (viscosity of the eddies). A correct calculation of the convective heat transfer coefficients at the walls depends on a suitable choice of the parameters of the turbulence model, adjusted under consideration of the weak natural convection. For the quality of the result, also the spacing of the grid near solid walls in combination with the choice of an appropriate turbulence model is very important. More accurate models (Large Eddy Simulation (LES) or Directly Numerical Simulation (DNS)) are only suitable for small models and not useful in large models with very different sizes of length e.g. a model of a cask storage.

In a CFD-calculation two different kinds of radiation-models can be used. The first one are surface to surface models, the second ones are ray tracing models, which are based on statistical calculations of the view factors.

The finned surface of the cask will be modelled as a smooth surface by using special fin models to consider the improved convective heat transfer. The first fin model is based on the Nusselt law $\text{Nu} = \text{C}\cdot\text{Ra}^{(1/3)}$, which holds for the natural convection at a vertical plate. An increase of the thermal conductivity near the solid surface within a finned zone is leading to higher heat transfer coefficients. The relation between surface enlargement and thermal conductivity can be developed analytically by the Nusselt law and the definition of the surface enlargement.

Another fin model is the roughness model. This model uses two parameters, the roughness height and the roughness parameter, which characterise the structure of the surface. The roughness height is chosen to be the height of the fins, whereas the roughness parameter has to be validated by experimental results.

5. Examples

In the following examples it will be shown that CFD calculation methods are suitable for a realistic prediction of the heat transfer and the temperature distribution at the surfaces of a cask and the cover.

Example 1: Validation of a roughness model for the finned surface of a CASTOR® V/19 cask

This example compares the results of the measured temperature at a CASTOR® V/19 with 20 kW standing within a concrete cover with a CFD calculation with FLUENT. The cover is furnished with inlet vents in the bottom and outlet vents at the top of the concrete side wall. The ambient temperature is 8 °C.

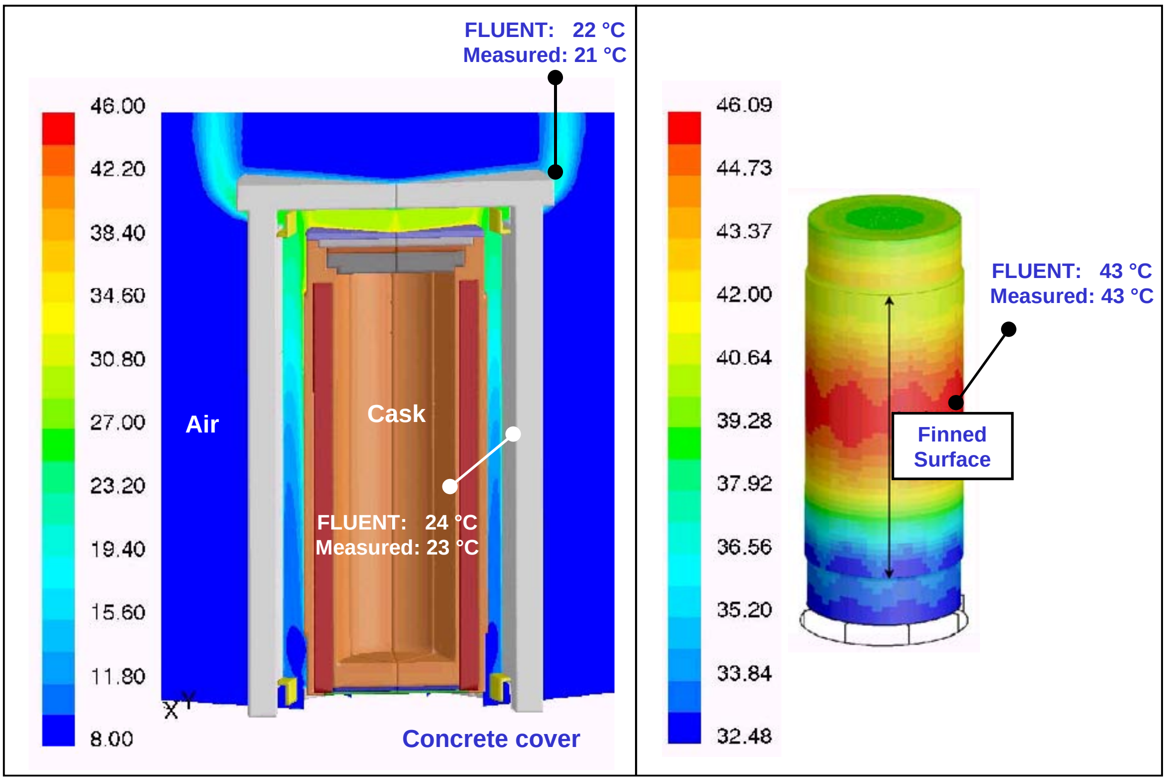

Figure 2: Measured and calculated temperatures at a CASTOR® V/19 within a concrete cover

For this calculation a 3D-model is used containing the cask, the concrete walls of the cover and the air within and outside the cover. The boundary conditions are applied considering the natural convection combined with the radiation between cask surface, cover and the environment.

The roughness height in FLUENT is set to 60 mm (height of the radial fins). The roughness parameter is adjusted in an iterative process, so that the maximum measured temperature and the calculated cask surface temperature are equal.

The calculation results with this roughness parameter are in a good agreement with the measured air and concrete temperatures. The differences of 1 K (see the temperature field in figure 2) are in the range of the accuracy of the measurement equipment.

Based on this result, further CFD-calculations are made by varying the roughness parameter to find a clear relation between the surface enlargement and the value of the roughness parameter. Those results are used for the calculation in the second example with many cask types with different surface enlargements.

Example 2: Validation of CFD for the real conditions in the storage facility in Gorleben

In this example the temperatures in the storage facility in Gorleben are measured with thermography and compared with the results of a CFD-calculation with FLUENT. Most of the casks (26) are of the type CASTOR® HAW 20/28 CG with a heat power from 27 kW up to 39 kW. The other 6 casks are of the type CASTOR® Ic, IIa, V/19 and TS 28V with a heat power from 2 kW up to 29 kW. The surface enlargements of the finned casks are between 2,6 (CASTOR® IIa) and 3,3 (CASTOR® HAW 20/28 CG). The finned zones of the casks are modelled as smooth surfaces in combination with the roughness model validated in example 1.

The calculation model consists of the concrete storage building with the in- and outlet vents, the several cask types and the air within the storage facility. The boundary conditions, which reflect the ambient conditions during the measurements, are 25 °C as ambient air temperature and about 150 W/m^2 for the solar absorption.

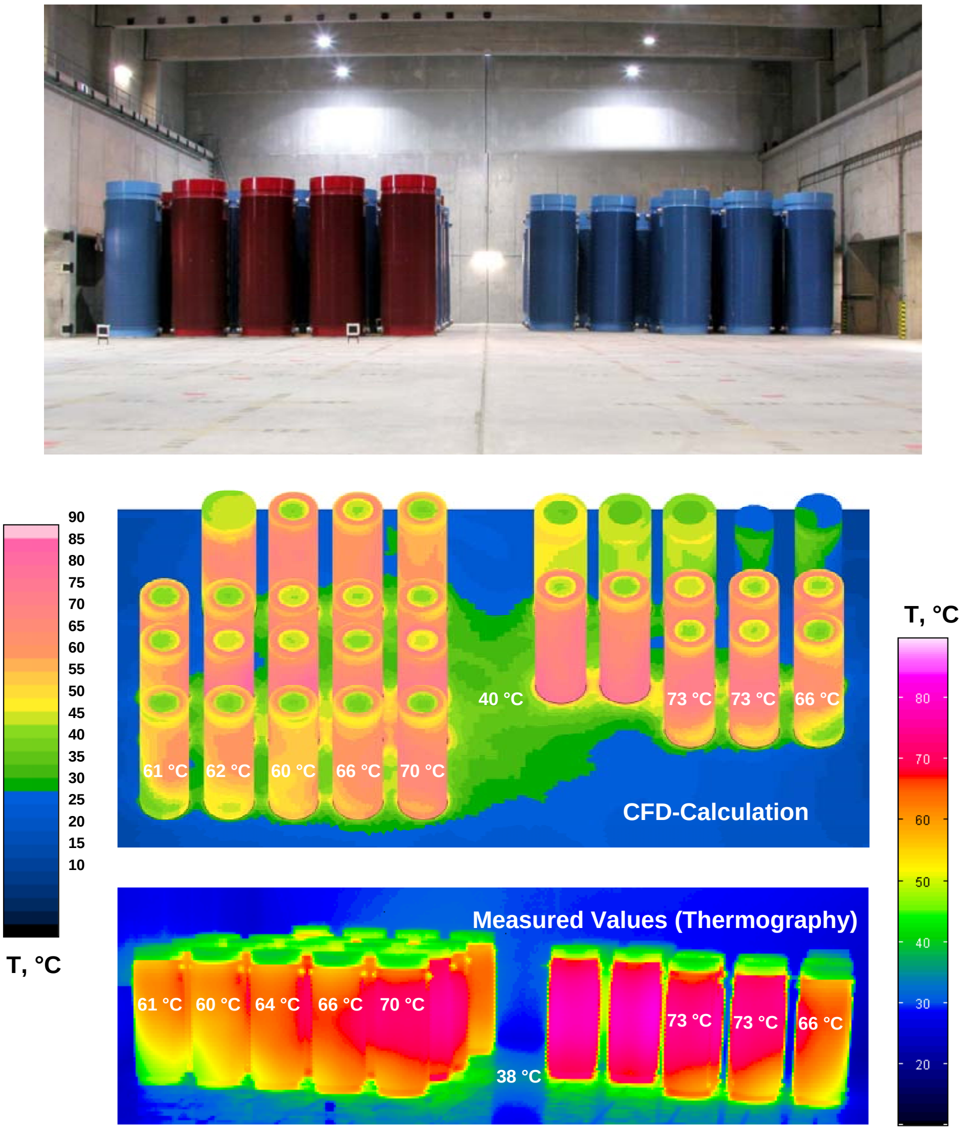

Figure 3: Measured and calculated temperatures in the storage facility in Gorleben

Considering the very complicated and asymmetric flow conditions in the space between the different cask types, the accuracy of the CFD calculation is quite astonishing. The temperature differences between the calculated maximum cask surface temperature and the measured values are less than 2 K, see figure 3. Also the calculated temperatures at concrete surfaces are in good agreement with the measured temperatures.

Example 3: Validation of CFD for a CASTOR® 20/28 CG lying under an optimised metal canopy

In this example the heat removal from the cask CASTOR® 20/28 CG with 38 kW lying under the optimised metal canopy of the type CH03 with additional outlet vents in the roof is considered. The measurements were made during the transport of the cask from La Hague to the storage facility in Gorleben. Therefore different measurement systems were used, including PT 100 thermo-elements, a thermography camera and a laser-pointer.

The CFD-model consists of the cask wall, the canopy and the air within and beside the canopy. The boundary conditions are chosen with respect to the ambient conditions during the measurements (12 °C ambient air temperature). The surface enlargement of the finned zone is applied by using the conductivity model, see chapter 4.

The flow field (see figure 4) and temperatures at the cask surface and the air temperatures calculated with ANSYS-FLOTRAN are in good agreement with the measured results. A comparison of the results is listed in Table 1.

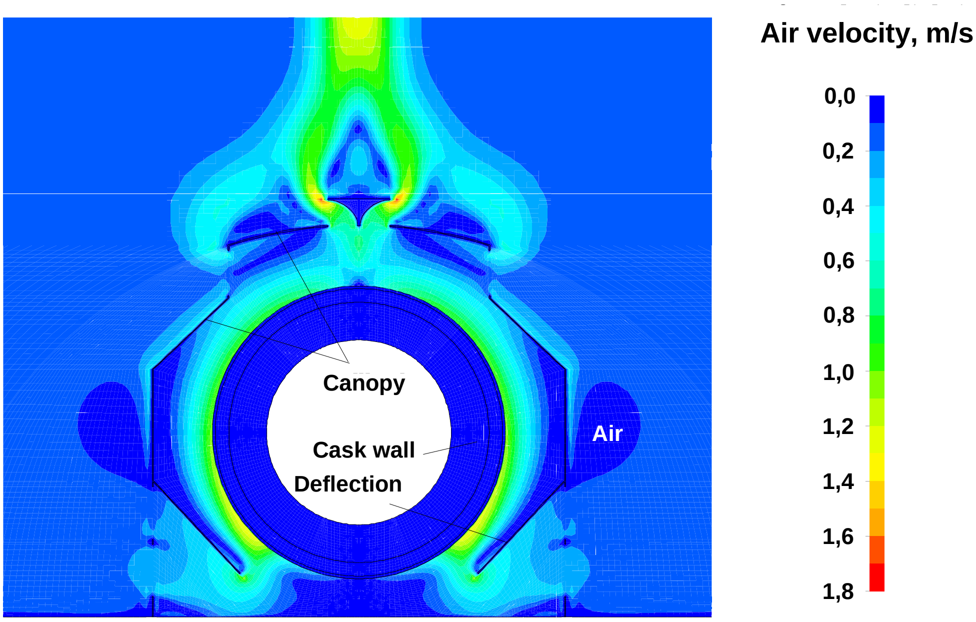

Figure 4: Air velocities under the optimised metal canopy

Cask temperatures, °C	**CFD**	**PT 100**	**Infrared**
Position			
Measuring area 1/2			
TB1/TB5	62.0/62.3	61.3/61.8	61.0
TB2/TB6	61.0/62.0	62.4/62.0	62.4
TB3/TB7	63.8/64.0	65.1/64.6	64.7
TB4/TB8	61.0/62.0	61.8/62.6	61.7
Average value	62.0/62.6	62.7/62.8	62.5
Air temperatures, °C	**CFD**	**Thermo-element**	
Position			
Inlet air TL1	15.5	14.3	
Inlet air TL2	15.5	14.4	
Outlet air (Roof)TL3	28.9	29.7 (half height of the opening)	
Outlet air (side wall)TL4	20.8	20.1 (half height of the opening)	
Outlet air (roof + side wall)	26.5	26.9	

Table 1: Calculated and measured temperatures under the optimised metal canopy

Example 4: Validation of CFD for a CASTOR® V/19 lying in an interim storage

In the next example the cask CASTOR® V/19 with 15 kW positioned under a thick cover of concrete during interim storage is considered. The calculation is performed with FLUENT using a 3D finite volume model. The CFD model consists of a very detailed model of the CASTOR® V/19 cask wall with all moderator bore holes and the concrete structures with in- and outlet vents.

The measured air temperatures given in Table 1 are used as a boundary condition in CFD. The surface enlargement of the finned zone is applied by using the conductivity model, see chapter 4.

The calculation results are in good agreement with the measured temperatures. Figure 5 shows that the ambient temperature averaged over 48 h is correct for estimating the maximum cask temperature, because of the thermal inertia of the storage unit (Casks and concrete cover). Higher air temperatures only present for a few hours will not affect the cask surface temperature. An additional non stationary calculation shows that an amplitude of ± 10 K during 24 h in the environment leads to a temperature amplitude of only 2.5 K at the cask surface.

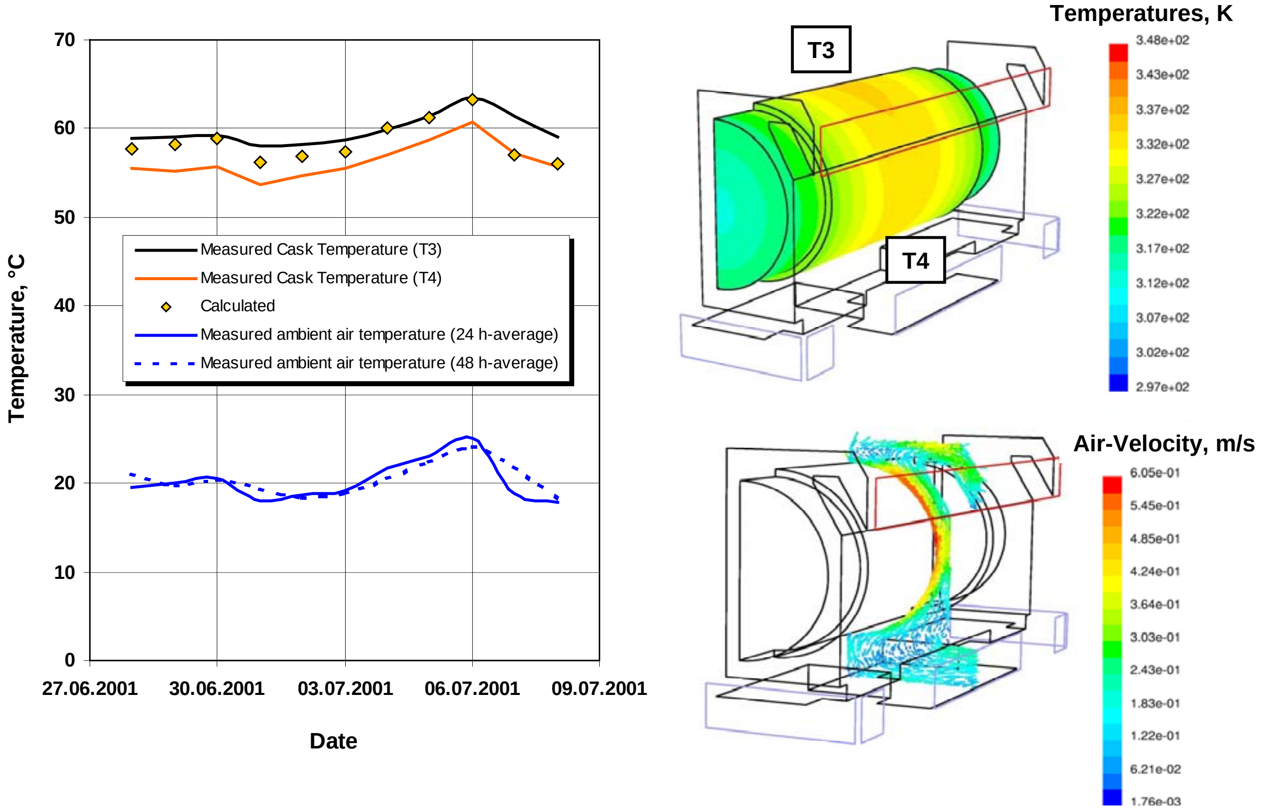

Figure 5: Calculated and measured results for the CASTOR® V/19 in the interim storage

6. Conclusions

In the validation examples, the numerical results of different CFD codes are compared with measured results at different GNB casks under transport and storage conditions. For the quality of the result, the spacing of the grid near solid walls in combination with the choice of an appropriate turbulence model is very important. If the axial heat transfer can be neglected, two dimensional models instead of three dimensional models are also possible.

The validation examples are leading to a very good agreement between the CFD calculation and the measured results. All calculation methods (turbulence model, radiation models, fin models) in different CFD programs used here (FLUENT, ANSYS-FLOTRAN) lead to realistic temperature predictions and are consequently suitable for the proof of safe heat removal for type B packages.