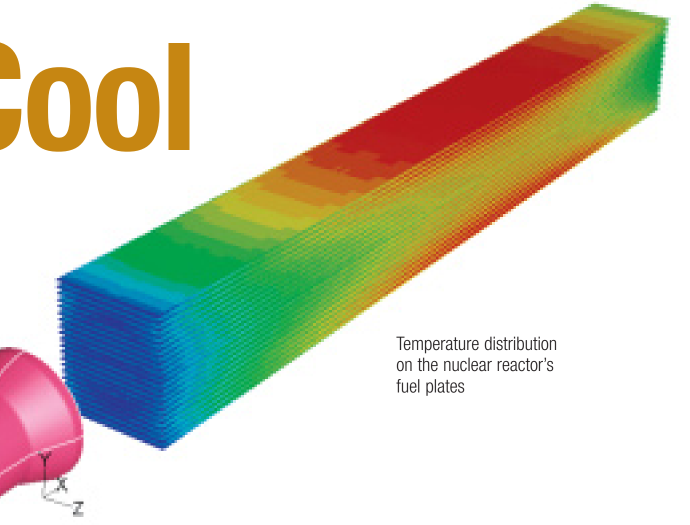
Keeping It **Cool**

Modeling fluid flow and heat transfer throughout a nuclear fuel assembly helps prevent reactor burnout.

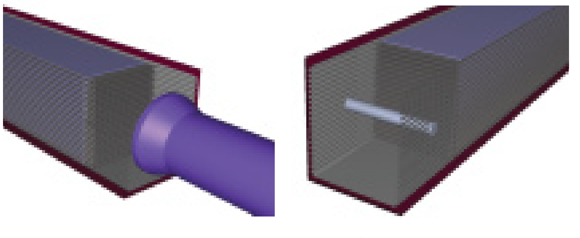
By Fahri Aglar, Turkish Atomic Energy Authority
Ankara, Turkey
Mustafa Ozer Gelisli and Emre Ozturk
ANOVA Ltd., Istanbul, Turkey



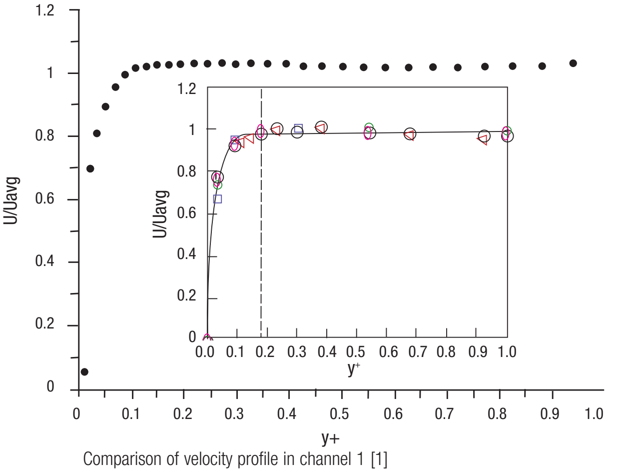
Temperature distribution on the nuclear reactor's fuel plates

Adequate cooling of fuel in nuclear reactors has always been an important safety concern. The bulk of the radioactive inventory of a nuclear reactor is contained in the fuel elements, and, normally, their integrity can be destroyed only by excessive temperature. Insufficient cooling of the fuel leads to burnout that can cause structural damage, and subsequent leaching of radioactive fission products. Therefore, the main goal of nuclear safety strategy is to avoid an imbalance between the heat generation and heat removal in all operational states. Such imbalance could result from transients in which either the heat generation exceeds the nominal values or heat removal falls below these values. Another cause of imbalance could be the loss of coolant from accidents that result in the partial or total depletion of coolant required for the heat removal. In past investigations of the problems encountered in cooling the fuel used in nuclear reactors, thermal hydraulic studies have been carried out both experimentally and theoretically [1, 2].

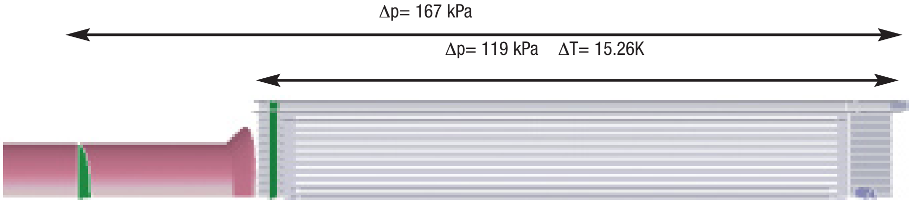
As part of its work studying reactor safety, the Turkish Atomic Energy Authority (TAEK) needed to evaluate the flow and heat transfer characteristics of a material test reactor (MTR)-type fuel assembly. As a provider of advanced engineering fluid mechanics solutions in Turkey, ANOVA Ltd. performed this study to assist TAEK in its evaluation.


In an evaluation of the safety of a nuclear reactor, ANOVA simulated the rotating and separating flow through the cooling channel and also modeled the wall shear stress and local heat transfer coefficients. Geometry created in GAMBIT 2.3 software of a material test reactor-type fuel assembly shows the diffuser and fuel plates (left) and outlet region (right).

The fuel assembly consisted of plate-type fuel elements, with light water serving as both the coolant and the moderator. Based on geometry and boundary conditions provided by TAEK, ANOVA generated a mesh of the assembly using GAMBIT software. By assuming symmetrical flow and geometry, only one-quarter of the fuel assembly needed to be modeled. When the cross section of the fuel assembly was examined, distinctive geometries with variable cross-sectional area — such as narrow cooling channels, slender fuel elements, and sudden enlargements and contractions — could be seen. Therefore, during the GAMBIT modeling, the fuel assembly was divided into three regions: the diffuser, the fuel plates and cooling channels between them, and the outlet region. A generally hexagonal mesh was developed, and the three regions were connected through non-conformal interfaces. Accurate evaluation of wall shear stress and local heat transfer coefficients at narrow cooling channels was required, which necessitated a boundary-layer meshing scheme. Under these circumstances, and following a sensitivity analysis, ANOVA analysts created a grid containing 2 million cells.



Comparison of velocity profile in channel 1 [1]

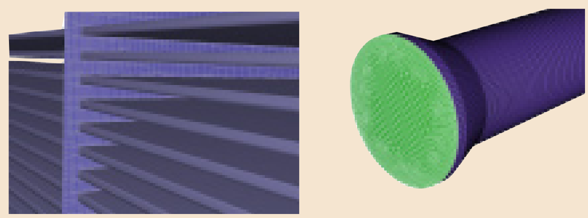


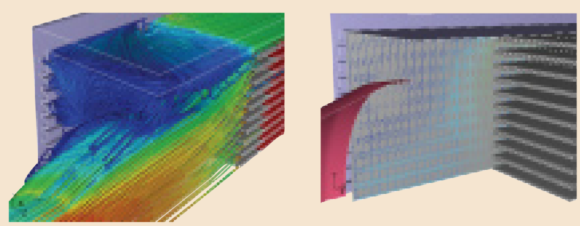
Pressure drop along the fuel assembly

The realizable k–ε turbulence model with standard wall functions was used throughout the computations to exploit its advantage for simulating flow possessing rotation and separation. The ANOVA team performed pressure and velocity coupling with FLUENT software using the semi-implicit method for pressure-linked equations, or SIMPLE. The convection and diffusion terms of the equations of motion were obtained by cell-based discretization. One of the main input variables for the FLUENT simulation was the volumetric heat generation, which TAEK extracted from the neutronic calculations using the WIMS-D/4 and CITATION codes. The power peaking factors, which describe the local power density at the hottest part of a fuel rod, also were estimated and used to correct the volumetric heat generation term.

One of the main concerns of the simulation was the comparison of the velocities at peripheral and central cooling channels. Engineers observed that the magnitude of the velocities at the peripheral cooling channels was slightly lower than the channels located at the center of the assembly. The reason for this became apparent when the influence of the vortex observed in a region between the diffuser and the fuel plates was taken into account. The vortex and its influence are extremely important from the reactor safety point of view, and estimations revealed that this velocity reduction seemed to be negligible. The outcomes related to velocity reduction also matched those obtained from experiment [1]. Further channel-to-channel flow distribution analysis showed that the relative flow rate, evaluated as a ratio of the flow rate in the individual channel to that of the assembly, decreased from the central channel to the outermost channel. The plate-to-plate temperatures showed the opposite behavior; that is, the temperature increased toward the outer channels.

A final point of interest was that the pressure difference between the inlet and the outlet of the fuel assembly was in the acceptable range and did not cause flow instability and phase change during normal operation. The pressure drop along the fuel region was 70 percent of the total pressure drop, which was in accord with experimental data [1]. The FLUENT results thus have been instrumental in understanding the complex 3-D flow in an MTR-type fuel assembly. Such CFD simulations have contributed significantly to the design and licensing of nuclear power systems. ■


Computational meshes in GAMBIT software for the cooling channel (left) and diffuser section (right)


Pathlines colored by velocity showing a vortex observed in the space between the diffuser and plate region (left) and corresponding velocity vectors (right)

www.taek.gov.tr
www.anova.com.tr

References

[1] Ha, T.; Garland, W. J., Hydraulic Study of Turbulent Flow in MTR-Type Nuclear Fuel Assembly, *Nuclear Engineering and Design*, 2006, 236, pp. 975-984.

[2] Franzen, F. L., Nuclear Power Plant Operational Safety — Safety Strategy and its Technical Realization, IAEA Interregional Training Course, Karlsruhe Nuclear Research Center, 1981.