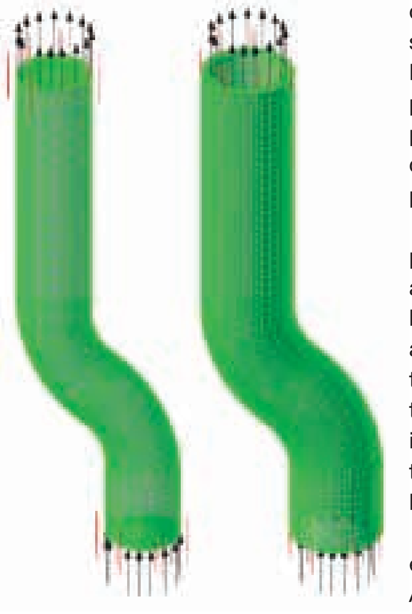
Image ©iStockphoto.com/Hans F. Meier

Coupling Analyses to Improve Nuclear Safety

Coupled thermal hydraulic and stress analysis of a CANDU feeder pipe helps determine integrity.

By Myung Jo Jhung, Principal Researcher, Korea Institute of Nuclear Safety, Daejeon, South Korea

The ultimate goal of nuclear safety regulation is to protect the public and the environment from the radiation hazards that could accompany the production and utilization of nuclear energy. The Korea Institute of Nuclear Safety (KINS) develops and implements nuclear safety programs, such as safety reviews and inspections, development of regulatory standards and monitoring of environmental radiation within Korea. In order to maintain and continually improve nuclear safety, increasing technology depth is required for prediction, analysis, experimental and remedial measures.



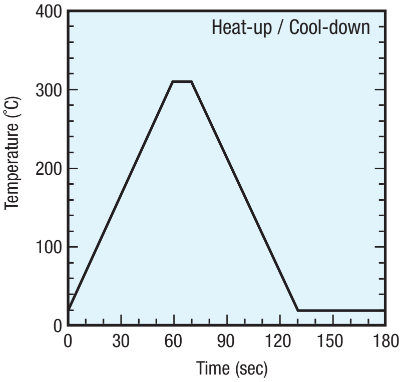
ANSYS CFX model of pipe exterior (left) and interior (right)

Because components in operating nuclear reactor systems can be subject to extreme forces and stresses that may threaten their integrity, safety is a constant concern. Safety is ensured by predicting conditions that would lead to component failures using simulations that incorporate fluid structure interaction (FSI) as a key technology. Simulations using FSI, for example, can involve taking results from a simulation of fluid flow with convective heat transfer and applying these results as loads in a structural simulation. In the past, these fluid and structural fields typically were analyzed separately due to the limitations of computer software and hardware resources. But advances in both areas now permit unified and efficient multiphysics simulations that couple the combined effects of interrelated physical phenomena (physics or fields).

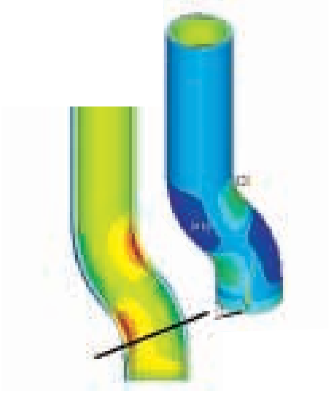
In this project, KINS researchers performed a coupled thermal hydraulic and stress analysis of a pipe with two bends. They studied transient heat-up and cool-down of the feeder pipe that delivers the primary coolant to the nuclear fuel of a CANDU pressurized heavy water reactor. The research team then used the results of this simulation for fatigue analysis of the pipe.

The team developed a finite element model for simulation using ANSYS Mechanical software. For this structural analysis, the engineers considered pressure and temperature simultaneously in generating the normal operating stresses. For the purpose of this study, a total time of 180 seconds was considered for heat-up and cool-down. Assuming an internal pressure of 10 MPa, the team of investigators discovered that maximum levels of equivalent stress and stress intensity were located in the intrados (inner curve) of the first and second bend. They also discovered that stress component variations along the circumference were more severe along the radius of the inner surface than along the outer surface.

The team used ANSYS CFX fluid flow simulation software to model the flow of the heavy water coolant and determine the temperature distribution within the heavy steel pipe. The investigators set the initial conditions to be



Transient thermal data representing a typical heat-up and cool-down cycle of the pipe model



Equivalent stress predictions in the pipe analysis assuming a constant interior pressure of 10 MPa. The line indicates where calculations for stress variations and stress variation components were conducted.

a stationary fluid and a temperature of 20 degrees Celsius for both the fluid inside the pipe and the pipe itself. As the heavy water flowed through the pipe, the temperature of the pipe increased due to the heat transfer between the pipe and the fluid. The team assumed a constant reference pressure of 10 MPa, and in their simulations included the variations of material properties with temperature of both the heavy water and the pipe.

KINS engineers then used the thermal results from the fluids simulation as input for a structural simulation that analyzed the resultant thermal stresses. They were able to obtain predictions of equivalent stress variations during heat-up (30 seconds) and cool-down (100 seconds). Analyzing the results and comparing the heat-up with the cool-down phases, the KINS team determined that the most severe axial and circumferential stresses arose at the outer surface during heat-up and at the inner surface during cool-down. As was seen with the pressure-based stresses, maximum thermal stresses occurred in the intrados of the bend.

The greatest thermal stresses found during cool-down, combined with the pressure-driven stresses, were used to determine the maximum equivalent stresses, which were quantified to be approximately 19 MPa. The fatigue curve for carbon steel [1] indicated a life of more than 106 cycles under this stress — much greater than what the feeder pipe is expected to see in operation. Therefore, the KINS researchers were able to conclude that the cumulative usage factor is almost infinite, and thermal fatigue of the pipe due to heat-up and cool-down over the time considered is negligible for this operating scenario.

Software from ANSYS allowed the KINS team to successfully perform a coupled thermal hydraulic-stress analysis of the CANDU feeder pipe to verify integrity estimates. By performing a unified simulation, the combined effects of the interrelated physical phenomena could be investigated efficiently, reducing both the time and the cost of independent simulations. At the same time, this approach provided a more realistic picture of the behavior of these components under the given operating conditions. ■

References

[1] ASME, ASME Boiler and Pressure Vessel Code, Section III, Appendix I, The American Society of Mechanical Engineers, 2004.

Simplifying the FSI Process with ANSYS Workbench

When integrating structural and fluids analyses, the intuitive interface of the ANSYS Workbench platform enables designers and analysts to account for one-way or two-way fluid-structure interaction (FSI).

For example, if an ANSYS mechanical simulation requires the results from an ANSYS CFX simulation in the specification of a load, users only need to select the relevant surfaces and the ANSYS CFX results file that contains the desired load. The ANSYS Workbench environment takes care of the rest, including management of files, extraction of data, interpolation between meshes and application of boundary loads.

User setup for two-way FSI only requires selection of the surfaces at which information such as temperatures or pressures are exchanged. The CFD and FEA solvers then run concurrently with robust implicit coupling on one or more machines connected by LAN, WAN or even Internet. Load transfer between the two uses an advanced algorithm that is both profile-preserving and conservative. There is no need for third-party software — all the data exchange is handled automatically and internally, using built-in socket-based inter-process communication (IPC).

There is no compromise in capability as the ANSYS FSI solution uses the full power and features of ANSYS CFX and ANSYS mechanical products.

– John Stokes, Product Manager
ANSYS, Inc.

Temperature distribution in the pipe for typical heat-up and cool-down. From left to right: 10, 30, 70, 100, 130 seconds, with blue indicating lower temperatures and red higher temperatures