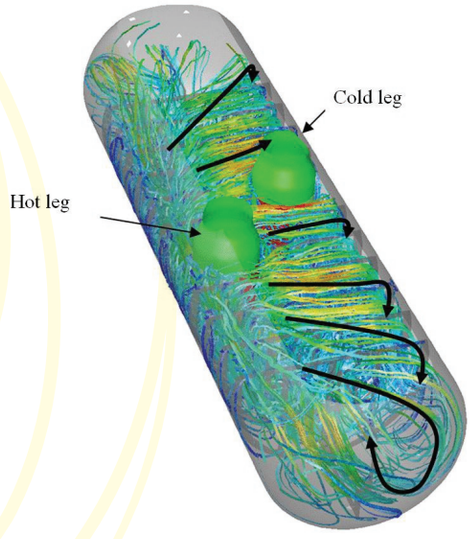
A New CFD Model of the Steam Generator for the VVER–440 Type Nuclear Reactors

Motivation

The inefficiency of the blow-down system is a known design problem of the VVER–440 type steam generators. For the prolongation of the lifetime of the Hungarian nuclear power plant in Paks the decided improvement of the blow-down system was conducted by the LG Energy Ltd. Since for the proper design of the blow-down system the contaminant concentration needs to be maximized at the extraction outlets, the analysis of the three-dimensional multiphase flow on the shell side of the tube bank had to be carried out. Our colleagues at CFD.HU Ltd. created a numerical model of the steam generator using ANSYS FLUENT simulation software with user defined functions (UDF) to enable this analysis. We are convinced that a useful model could be developed along the same lines for the VVER–1000 type steam generators as well.

How can the model of the steam generator help you?

- Treats the steam generator as a distributed parameter system
- Provides full, three dimensional and time-dependent, picture of the flow and other state variables
- Capable of handling geometrical variants
- Useful in comparing the effects of different operating conditions
- Useful to analyse individual tube plugging configurations
- Enables to study operational transients
- Capable of tracking particulate contaminants and the dispersion of solved chemicals
- Helps to identify locations of corrosion hazards and clogging



Components of the steam generator model

- A detailed geometrical model to resolve the interior structure elements of the steam generator
- Porous zones to describe the anisotropic and inhomogenous shell-side resistance of the tube bank
- A coupled 1D thermo-hydraulic model of the primary side to uncover the unevenness of heat-exchange
- The mixture model of ANSYS FLUENT is employed to describe the coupled process of pre-heating, condensation and heat-exchange
- A model to specify the local value of slip velocity of water vapour relative to the liquid
- Hydraulic models to include feed water intake, blow-down extraction and water level control

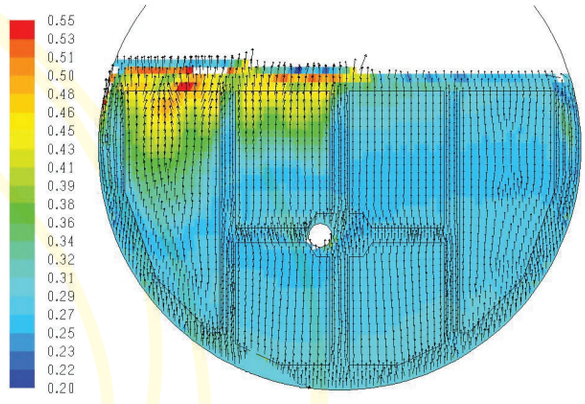
Methodology details

Geometry and mesh: Two different resolution 3D meshes were generated: a fine one with 1.2 million cells, and a simplified one with 0.5 million cells. The latter was used only to relax artificial initial transients and thus shorten computational time. A 2D mesh was also generated which was very useful for physical model development.

Feed water intake: Two geometrical variants of the feed water intake system were modelled: both low level injection (the original design) and near-surface injection (the present operation in Paks) can be alternatively activated in the simulation. The spatial distribution of the incoming feed water was calculated from hydraulic models of the actual intake layout in both cases. The model has the ability to incorporate other variants that are in operation in other plants.

Hydraulic resistance: The tube bank of the steam generator has been modelled as an inhomogenous and anisotropic porous medium, the permeability matrices of which are obtained zone by zone based partly on 2D micro-models of the flow.

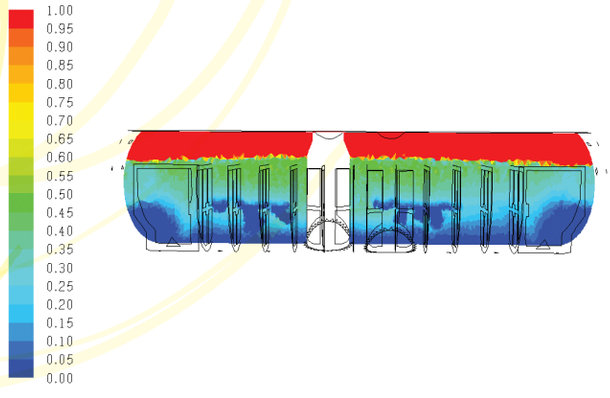
Heat exchange: Heat flux through the heat exchanger tubes and the consequential increase in the specific volume of the fluid in pre-heating and evaporation induce the shell side water circulation. Local heat source density has been computed using coupled 1D simulation of every tube (in collaboration with LG Energy Ltd) based on a hydraulic model of the primary side. Potentially, the exact locations of inactive (plugged) tubes can be taken into account in the model as well.



Working fluid properties: The secondary coolant fluid was treated as a mixture of saturated vapour, saturated water and a fictive third phase, the feed water. The mixture model of ANSYS FLUENT has the ability to describe the physical processes of two-way phase transition and heat transfer among the phases.

Slip velocity: The mixture model needs to be supplemented by an adequate slip velocity model for the realistic formulation of the relative motion of the liquid and steam phases. A semi-empirical correlation for kettle reboilers was adapted for this system by taking into consideration the vector sum of gravity and inertial accelerations and the known characteristics of the wet steam at the outlet.

Unsteady simulations: Time dependent simulations have been performed in every case. This enables the modelling of transient behaviour during varying operating conditions.

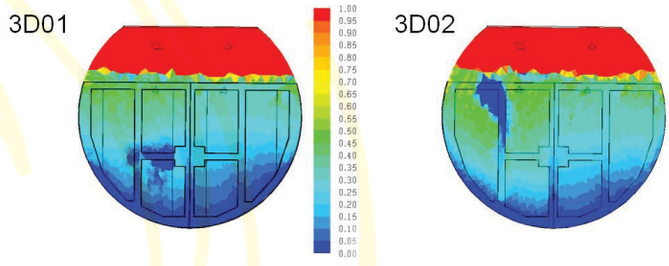


Numerical control of water level: In the actual technological process water level is controlled by real time signals of water level measurements. This feedback mechanism is modelled by a numerical level control.

UDF modules: Almost all the above mentioned models were at least partly programmed on the source code level taking advantage of the user defined function capabilities of ANSYS FLUENT.

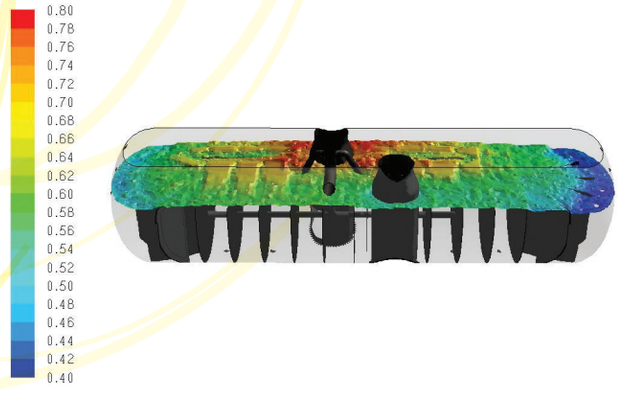
Results and conclusions

With the help of the complex model of the steam generator, the shell side circulation was visualized and the unevenness of steam production was also explored. The major flow pattern is almost two dimensional, perpendicular to the container axis, in most part of the steam generator. Markedly three dimensional flow structures occur only at the end walls of the vessel and in the vicinity of the collectors.

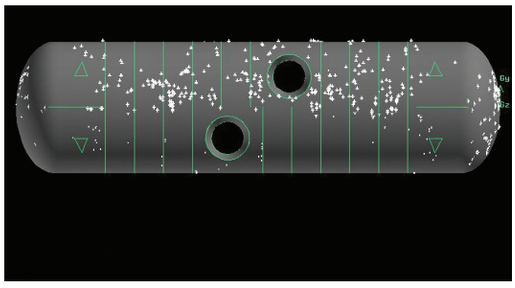


Comparing the results of the simulations with low level and near surface injection variants indicated that the feed water intake system has a large impact on the shell-side circulation: traces of low temperature feed water plumes can be clearly identified in Figures 3D01 and 3D02.

A well defined water surface is formed, above of which the volume fraction of vapour approaches 100%. The surface shape obtained from the simulation model is in good correlation with earlier experimental observations. [Reference: Dmitrijev A. I., Kozlov Yu. V. et. al.: "Experience with the Application of Venetian-Blind Separators in Atomic Energy Station Systems" (in Russian), Teploenergetika, Moscow, 1989]



The numerical simulation provided a complete picture upon the spatial distributions of velocity and void fraction. This allows further analysis of the transport of possibly corrosive contaminants in the vessel, which is essential for improvement of blow-down efficiency.



Lagrangian particle tracking analyses showed that the sedimentation of disperse particles occurs mainly below the hot side straight tube segments close to the stems of separation vortices we identified on the shell bottom in agreement with the operational experience.

The global flow field obtained from the steam generator model can be used to provide realistic boundary conditions for CFD micro-models of certain details of the system. One such micro-model revealed the existence of detached separation zones in the narrow gaps of the tube support structure. The locations of such badly rinsed regions correlate well with the known statistics of positions of actual tube corrosion indications.

About us

CFD.HU Ltd. is a computational fluid dynamics consultation company closely associated with the Department of Fluid Mechanics at the Budapest University of Technology and Economics (BME) and the distributor and support centre for ANSYS simulation software. With the support of market-leading ANSYS Fluid Dynamics and Mechanical analysis software, the company's internationally acclaimed staff can analyse complex thermal and fluid mechanical problems in order to optimise future engineering designs.

The company is co-lead by Dr. Tamás Lajos, the former Head of the Fluid Mechanics Department, and Dr. Gergely Kristóf, the chairman of the sub-committee of Fluid Mechanics and Heat Technology at the Hungarian Academy of Sciences (MTA). Their colleagues have a rich academic background with years of experience in the field of computational fluid dynamics. The company's close relationship with the University also provides the opportunity to perform wind tunnel investigations and other experiments to validate CFD results.

Numerous further application of the CFD steam generator model is foreseeable, for further information visit www.cfd.hu where you can find an essay about the investigations, or e-mail us at info@cfd.hu.

Address: Hungary 1117 Budapest, Gábor Dénes u. 2. INFOPARK building D