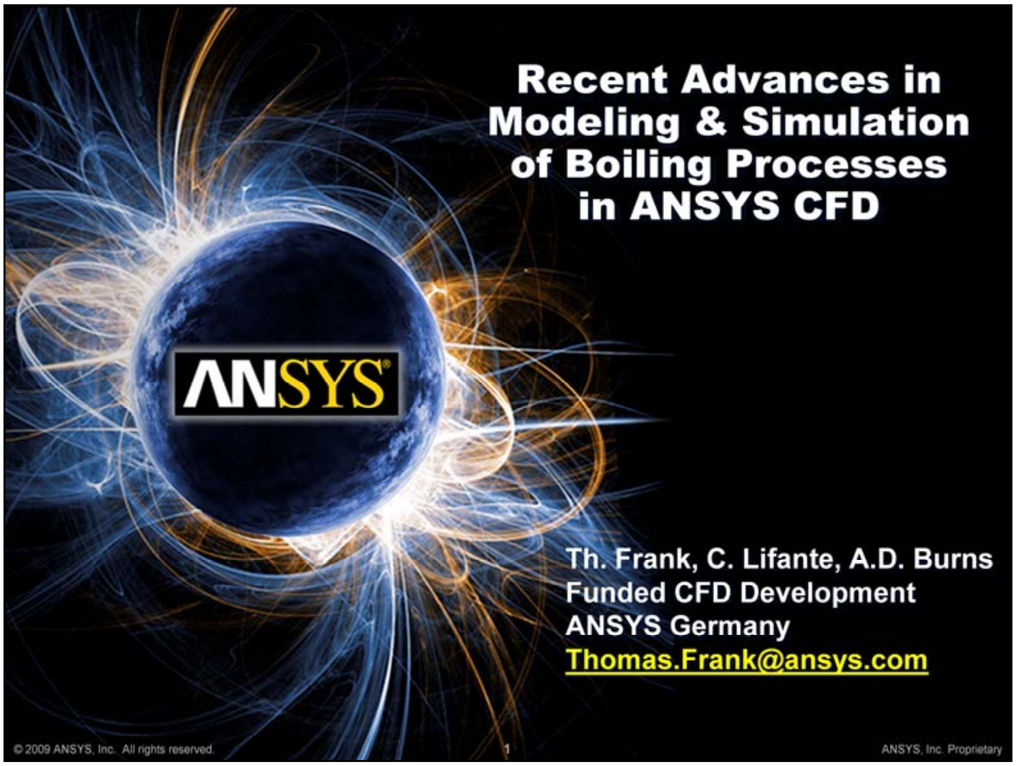
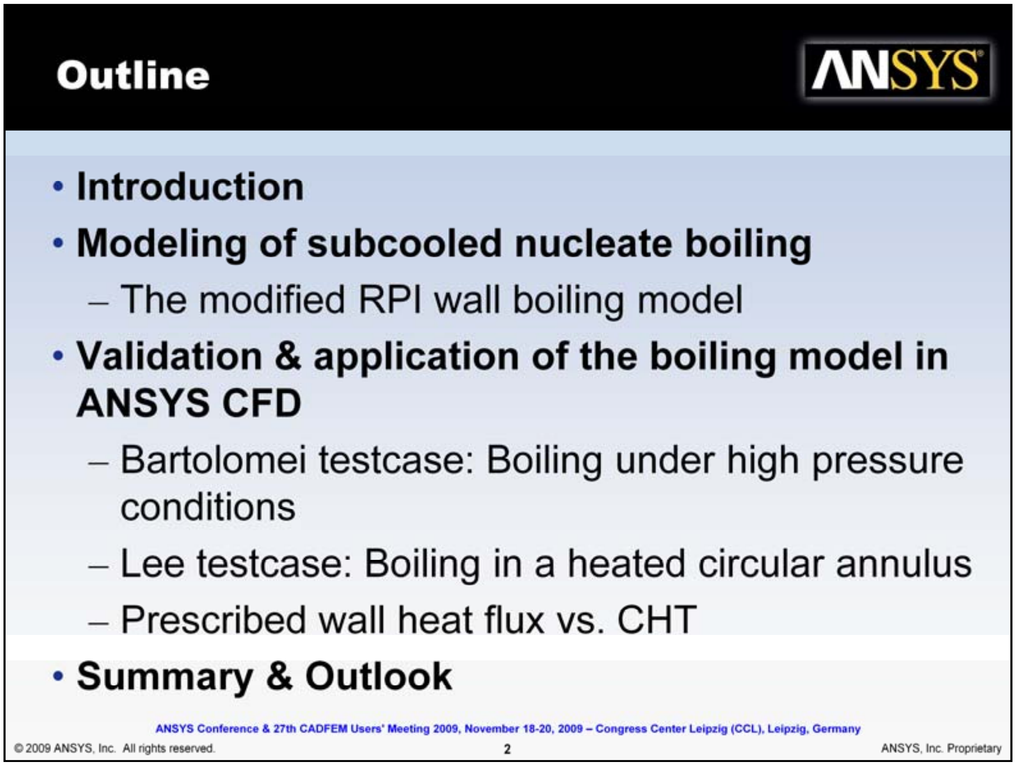
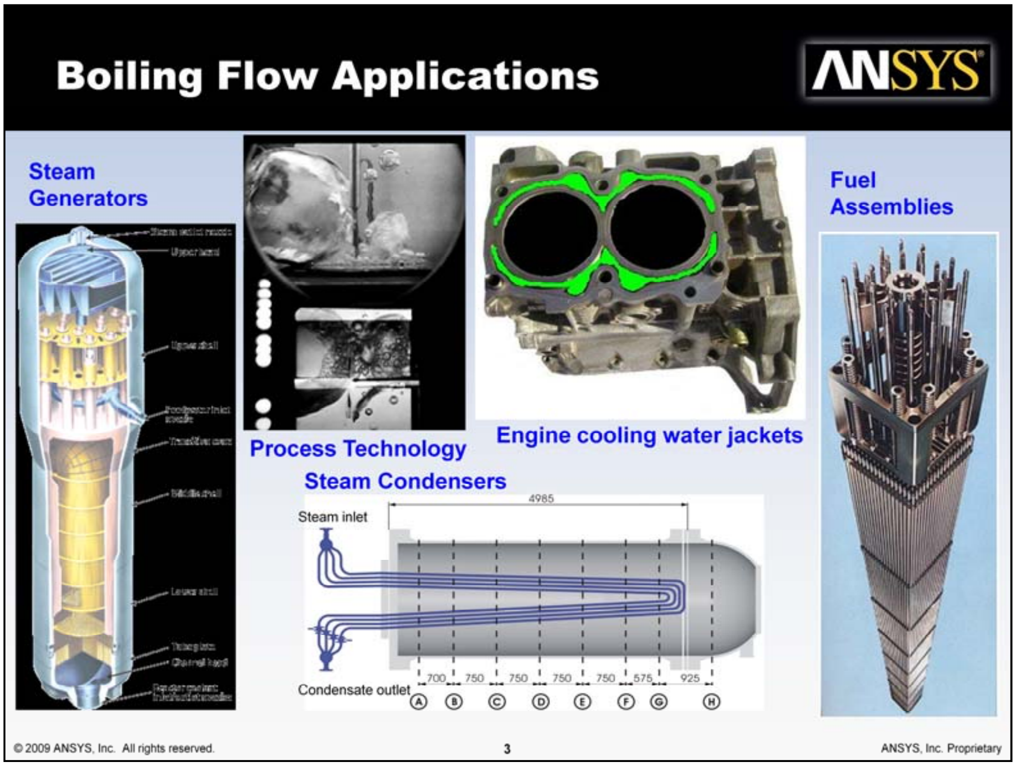

Recent Advances in Modeling & Simulation of Boiling Processes in ANSYS CFD
ANSYS
Th. Frank, C. Lifante, A.D. Burns
Funded CFD Development
ANSYS Germany
Thomas.Frank@ansys.com
© 2009 ANSYS, Inc. All rights reserved.
1
ANSYS, Inc. Proprietary



The main focus of this paper is to present recent advances on the implementation, validation and application of the so-called RPI wall boiling model for nucleate subcooled boiling flows. Besides some details of the model formulation the paper will cover 2 investigated testcases:

1) The testcase of Bartolomei et al.; nucleate subcooled boiling in circular pipe with heated walls under high pressure conditions
2) The testcase of Lee et al.; nucleate subcooled boiling in a circular annulus with a centralized heated rod on the symmetry axes; boiling under almost ambient pressure conditions, so low pressure

Furthermore the presentation will show the application of the RPI wall boiling model together with heat conduction prediction in the solid material of the heater (CHT). Finally the paper will give some outlook to future intended ANSYS R&D on modeling of boiling processes.



Boiling processes in relevant flow simulation scenarios are wide spread in industrial applications. Some examples of boiling flows are:

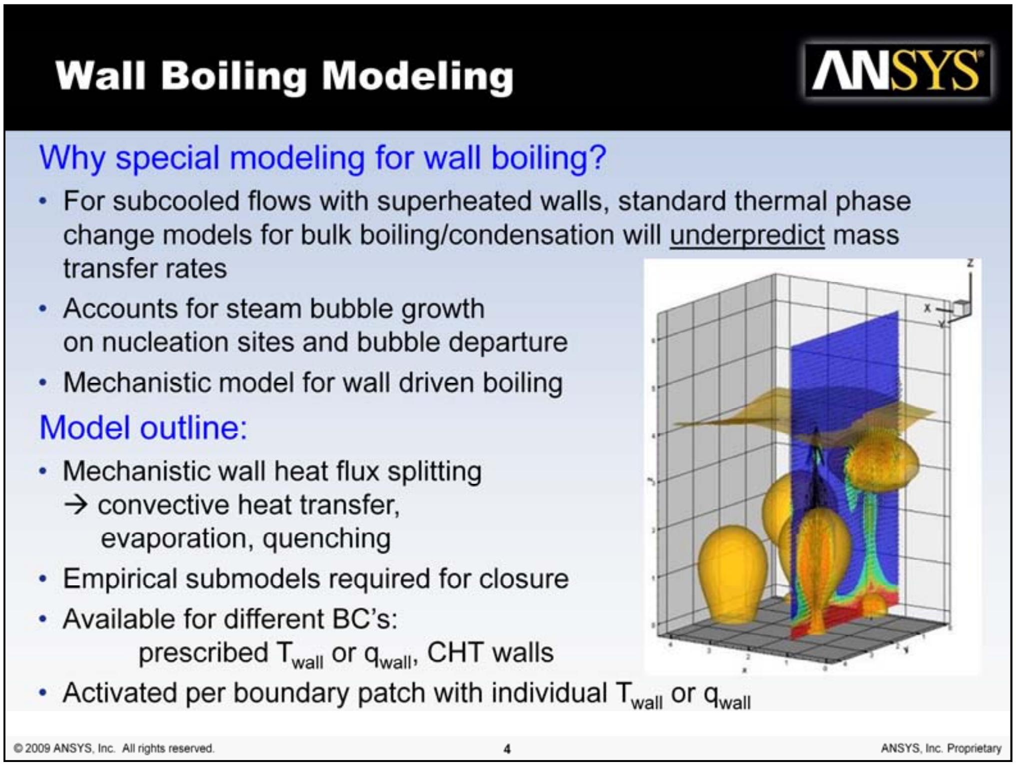
• Boiling in steam generators in conventionally fueled and nuclear power plants

• Boiling in the fuel assemblies of PWR and BWR nuclear power plants

• Boiling which might occur under certain circumstances on the rod bundles in large steam condensers

• Local wall boiling in the cooling water jacket of internal combustion engines of motorcycles, cars as well as large ship Diesel engines. Usually such local boiling in ICE has to be avoided, since boiling comes with steam generation and therefore with a large increase of fluid volume. Sicne cooling water jackets are in most cases closed systems, boiling has not to occur.

• Boiling in process technology and chemical engineering processes, where strong heating is e.g. required in order to facilitate certain chemical reaction.



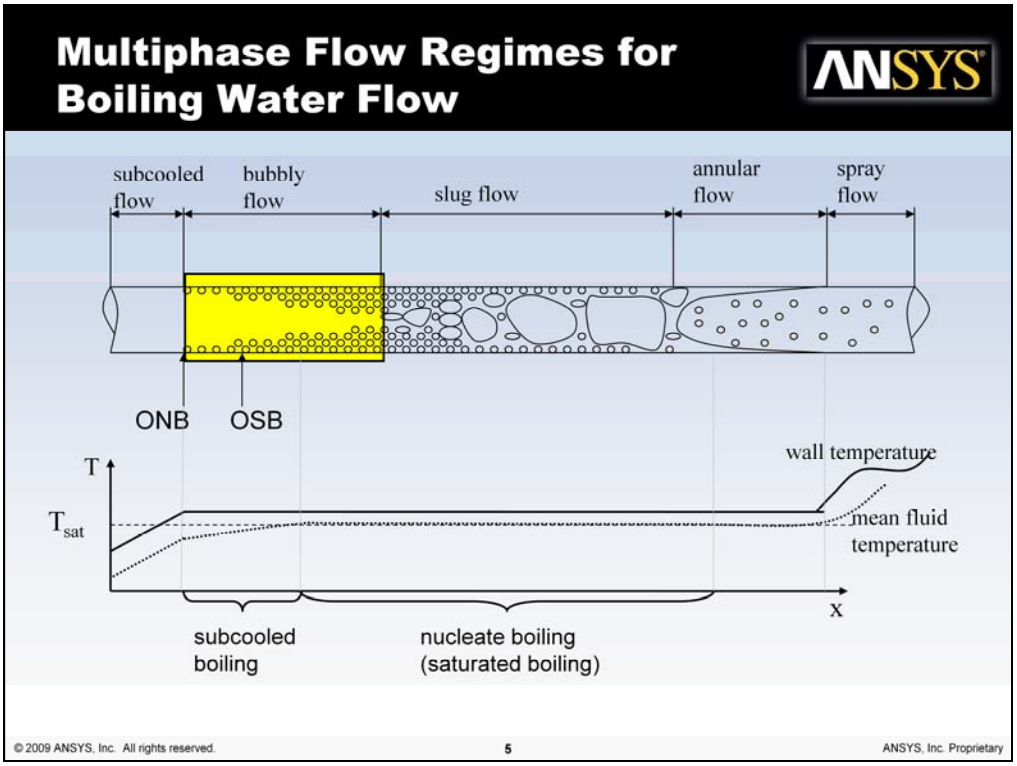
Why we need a special wall boiling model? If we have a model which accounts for heat and mass transfer in the bulk multiphase flow if the liquid phase temperature exceeds saturation temperature – isn't that enough to account for the boiling, if we bring the heat from the wall into the liquid by convective heat transfer at the wall?

The answer is no. The so-called bulk boiling model (or thermal phase change model) would substantially underpredict the steam production, especially for subcooled boiling, where the liquid temperature has initially a temperature of several Kelvin below the saturation temperature (liquid subcooling). Application of a pure bulk boiling model would substantially delay the onset of boiling.

The wall boiling model has to account for the early steam bubble growth on nucleation sites directly on the surface of the heater, later bubble departure and enhanced heat transfer from the heater surface by evaporative and quenching heat fluxes (so not only convective heat transfer as in a single-phase flow).

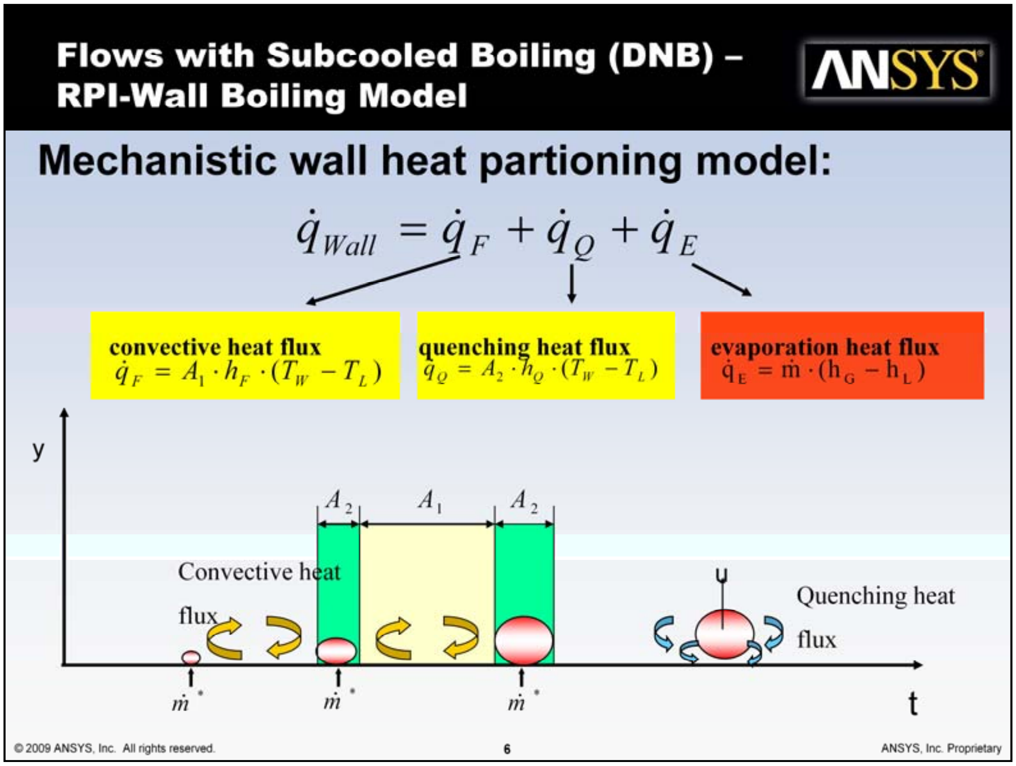
The established wall boiling models account for these additional components of the heat flux from the heater surface into the multiphase flow mixture. but due to the complex physics of the boiling process and the limited spatial resolution of an Eulerian multiphase flow CFD simulation, some underlying physical phenomena and processes cannot be resolved on the scales of the numerical mesh and have therefore subsequently been modeled by mechanistic submodels, empirical model closures relying on experimental investigations.

The modified RPI wall boiling model available in ANSYS CFX can now be used with different types of boundary conditions: prescribed wall temperature, prescribed wall heat flux or in combination with resolved heat transfer in the adjacent solid domain of the heater (CHT). Furthermore the model has been implemented that it can be activated on a per boundary patch location basis. So different boundary conditions and different model settigns can be specified for different wall areas, e.g. different nucleation site densities and bubble departure diameters for different heated wall materials (different metals, different state of corrosion, different manufacturing, etc.).

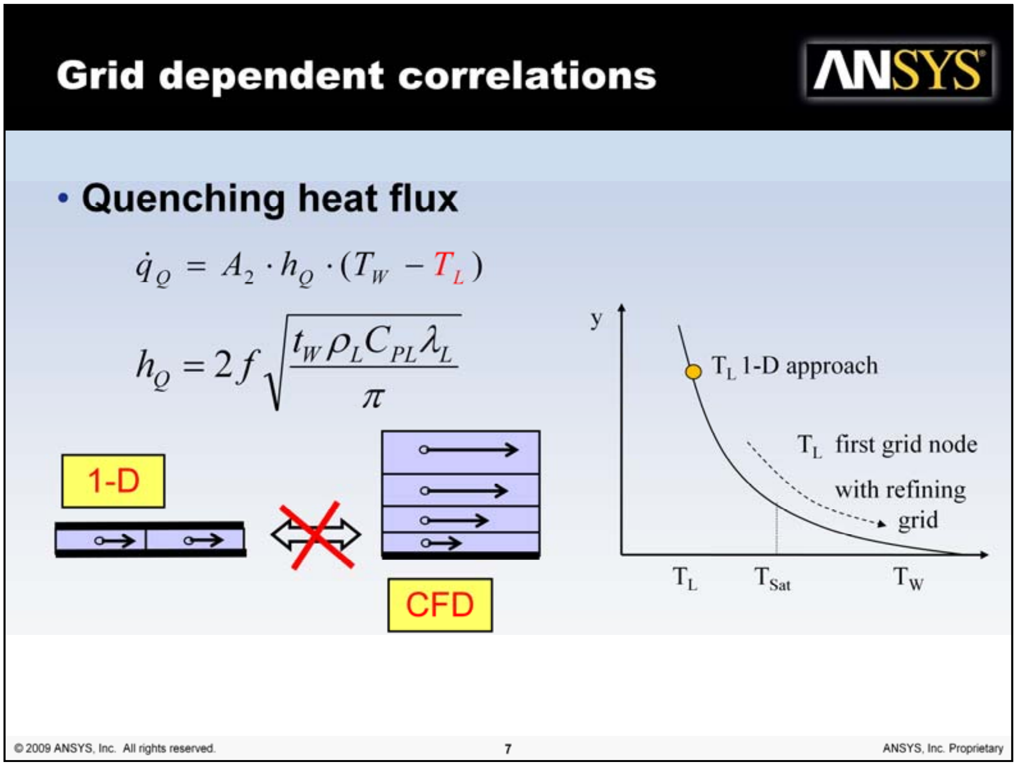


If we consider flow conditions in a pipe or channel with heated walls, then we observe a change from single-phase subcooled liquid flow, to bubbly flow (ONB – Onset of Nucleate Boiling, OSB – Onset of Significant Boiling), slug flow regime with nucleate boiling, annular flow and finally the formation of droplet flow under dry-out conditions. The lower schematic diagram shows the behavior of wall and mean fluid temperature in comparison to the fluid saturation temperature in correspondence to the changing flow regimes.

The modified RPI wall boiling model is – strictly speaking – only applicable to the bubbly flow regimes of nucleate boiling up to DNB (departure from nucleate boiling).



The Rensselaer Polytechnical Institute (RPI) first developed the so-called RPI wall boiling or heat partitioning model. In this model the overall heat flux from the heated wall to the two-phase flow (the subcooled liquid with steam bubbles) is divided into 3 parts: a convective, quenching and evaporation heat flux. Furthermore the heat flux partitioning model associates each of the heat flux contributions with a dimensionless wall area ratio in order to define the ratio between heat flux contributions.



Originally the RPI wall boiling model has been developed for 1-dimensional flow modeling and relates the convective and quenching heat flux contribution to the bulk liquid temperature. But in the framework of a CFD algorithm this value is locally (at the wall nearest mesh cell) not available. If the required liquid temperature value is nevertheless taken from the wall-nearest grid cell, then the model becomes grid dependent and inaccurate and the quenching heat flux will reduce with increased near wall resolution. Thereby the heat flux partitioning becomes inaccurate and overpredicts the evaporation and convective heat fluxes.

Grid dependent correlations

ANSYS

- **Evaporation heat flux**

$$\dot{q}_E = \dot{m} \cdot (h_G - h_L)$$

$$\dot{m} = \frac{\pi \, d_W^3}{6} \rho_G f \, n$$

$$d_W = \min\left\{1.4mm, 0.6\,\mathrm{mm} \cdot \exp\left(-\frac{T_S[\mathrm{K}] - T_L[\mathrm{K}]}{45[\mathrm{K}]}\right)\right\}$$

d_W – bubble departure diameter

n – nucleation site density per m²

f – bubble departure frequency

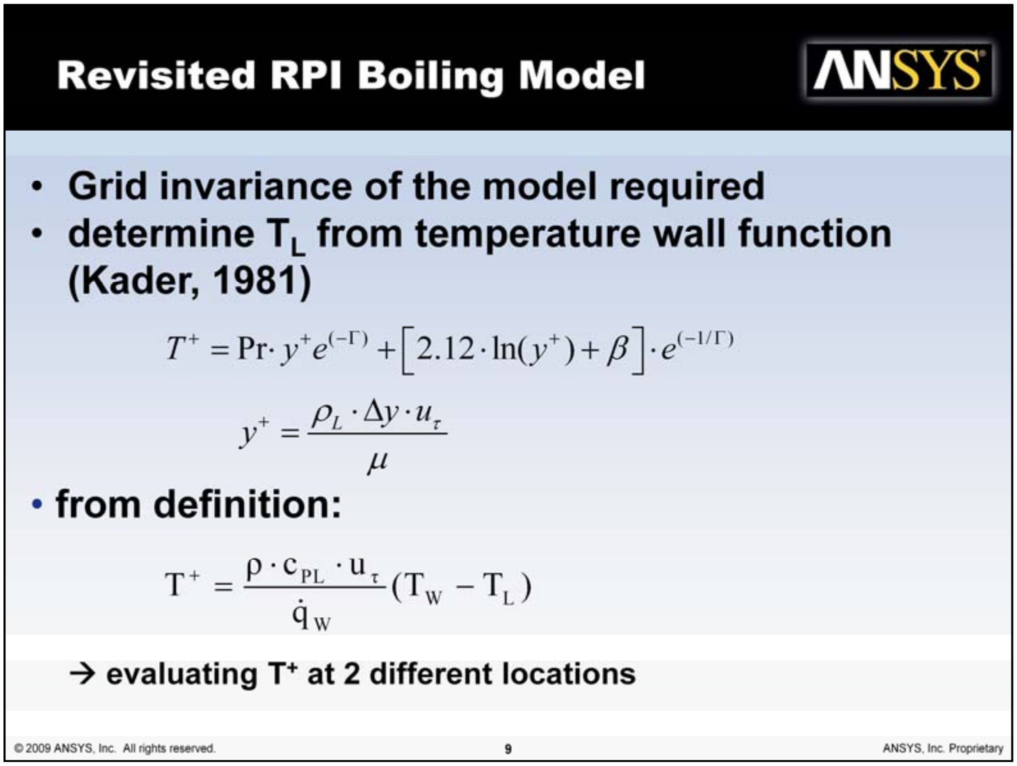
small quenching & overestimated evaporation on fine grids

wrong heat flux partitioning

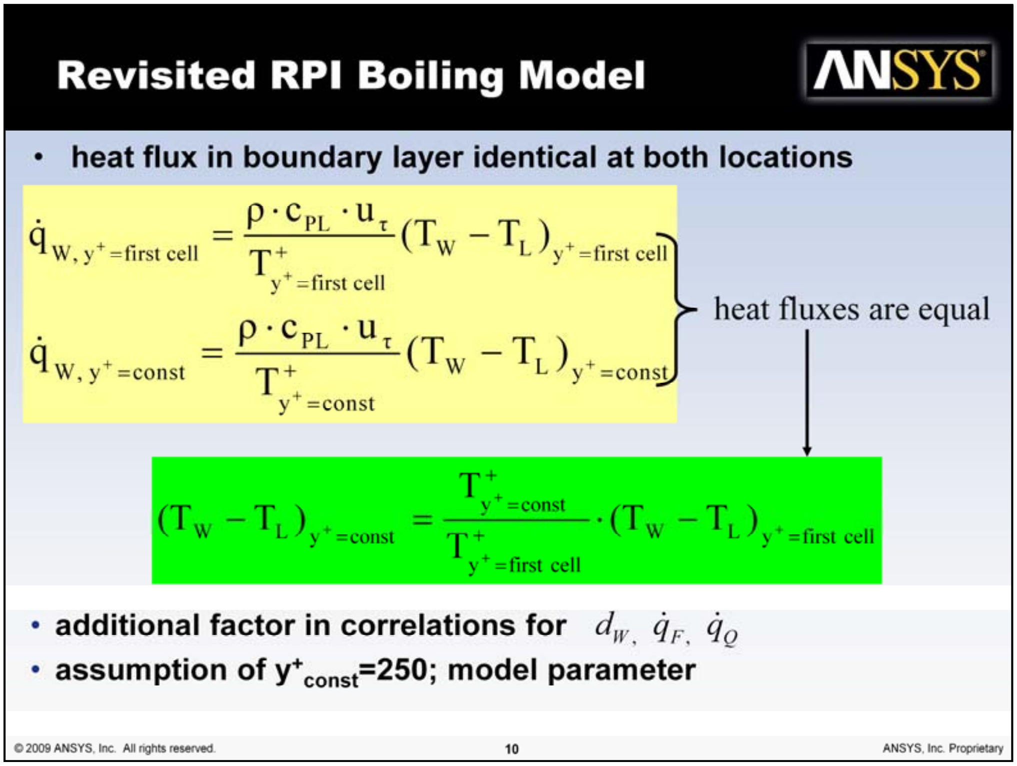
→ tends to film boiling on fine grids (due to $T_L \rightarrow T_W$)

 8

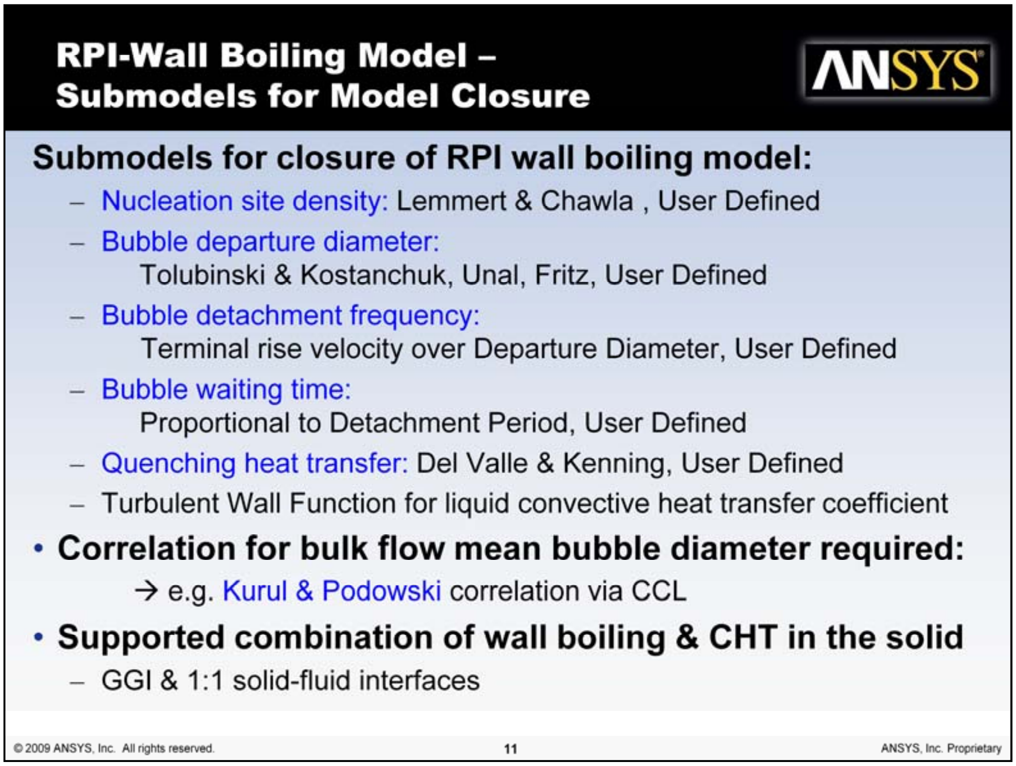
The same issue appears in some of the closure correlations of the model, e.g. for the bubble departure diameter used in the evaporation heat flux. The use of the wall nearest grid cell value of the liquid temperature instead of the non-available bulk liquid temperature leads to the tendency of too high vapor production and therefore to film boiling.



In ANSYS CFX the wall boiling model has been revisited and consequently modified for the needs of a grid independent implementation into a CFD code. The determination of the near wall liquid temperature was based on the temperature wall function of Kader (1981) and by evaluating T+ at two different locations.



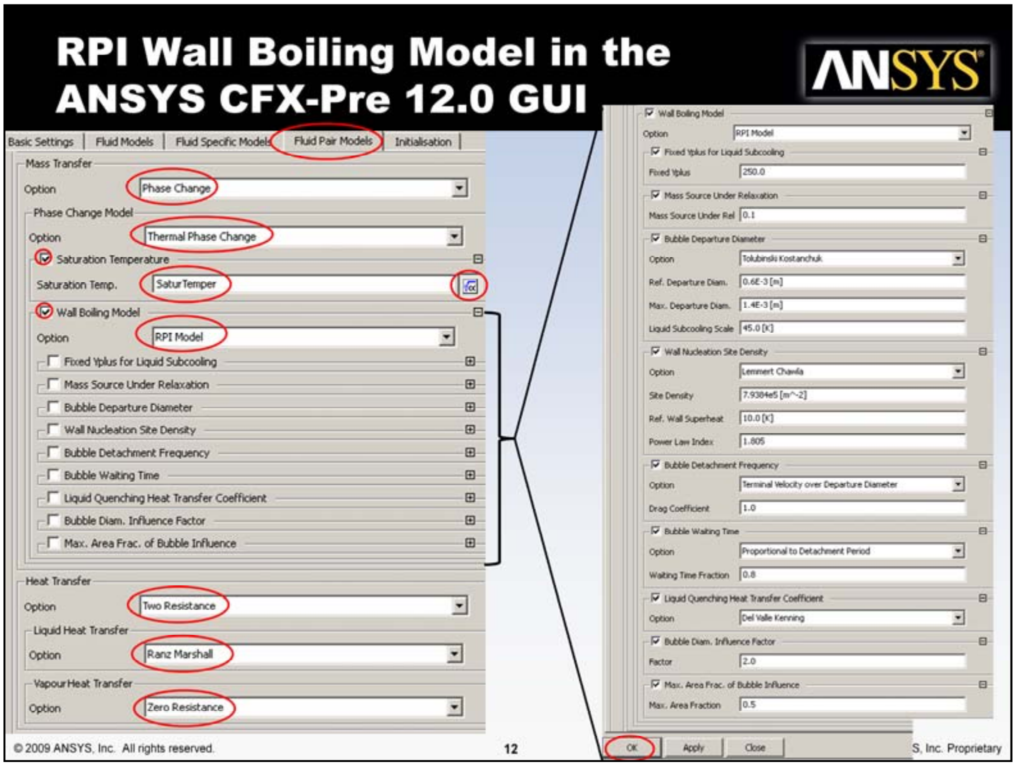
Since the heat flux in the boundary layer should be equal for both evaluated wall distances (in the first grid cell and at a constant y+), the resulting two heat fluxes from the above expressions can be equalized. From the resulting equation we can now determine the difference between the wall temperature and the bulk liquid temperature in dependency on the given values of the wall temperature and the liquid temperature in the wall nearest grid cell. An additional pre-factor occurs in this relation. The wall distance of heat flux evaluation is a model parameter and was set to y+=250.



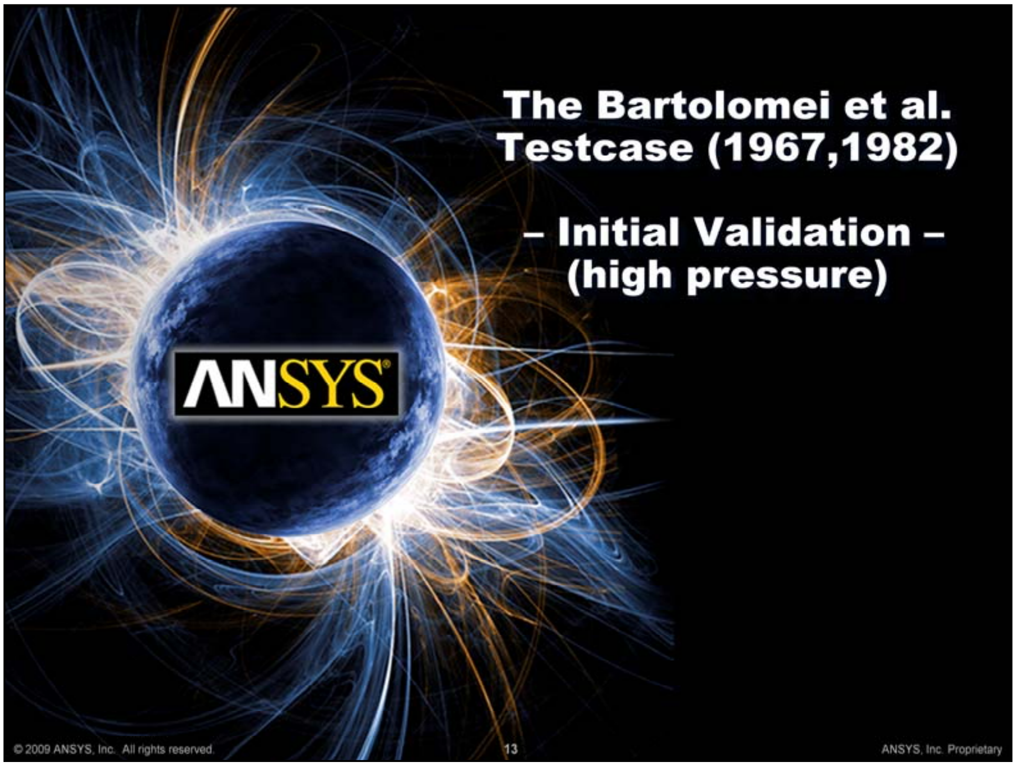
Due to limited mesh resolution in a CFD simulation in comparison with the microscopic length scales of the wall boiling process we need to apply empirical closure for some underlying physical process. The required closure models for the wall boiling model are listed on this slide. For most of them the most popular correlations from the open literature had been implemented in ANSYS CFD and are provided to the user. Further correlations can be implemented by users themselves using either CCL or CEL User Fortran functions.

besides submodels of the wall boiling model another important submodel for boiling processes in multiphase flows is the information about the local bubble diameter in the bulk flow and thereby information about the interfacial area density. The latter is important influence factor for any heat, mass and momentum transport between phases. For rather unidirectional flows, e.g. in subchannels of nuclear reactor fuel assemblies, it is common practice to use for this bulk bubble diameter correlation based information, which relates the bulk bubble diamter to the local liquid subcooling temperature. But these correlations might be flow condition and pressure level dependent. Therefore for future development it is intended to couple the wall boiling mdoel with some kind of population balance model like DQMOM or inhomogeneous MUSIG models in order to replace this correlation based information by more predictive CFD methods.

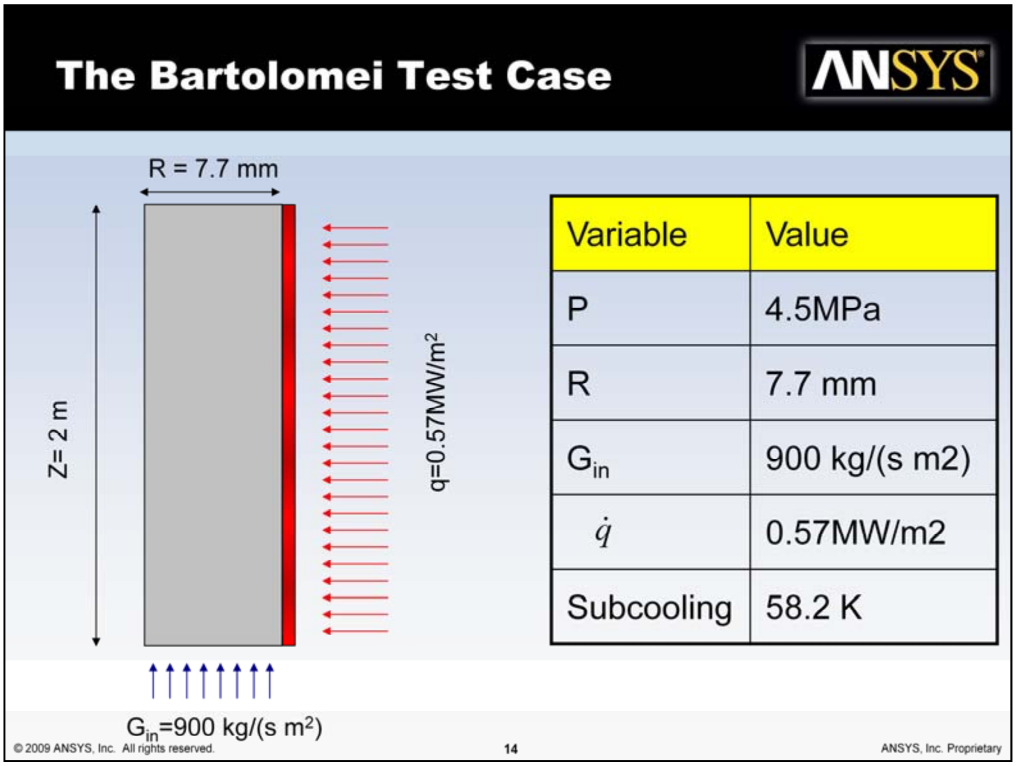
Another important feature is the coupling of the modified RPI wall boiling model with CHT. The assumption of a constant wall temperature or heat flux does not hold in many applications. Therefore it is desirable to predict 3d heat transfer and temperature distribution in the solid material of the heater as well and thereby predicting the heat flux to the fluid domain more accurately. With the release of ANSYS 12.0 this goal has been realized.



The slide shows the GUI of ANSYS CFX5Pre 12.0 for the specification of wall boiling model parameters and submodels. Most commonly used submodels are provided to the users as selectable options from this GUI. Further submodels can be brought into the CFD simulation by CCL and CEL user Fortran functions.



Initial model validation on the Bartolomei testcases from 1967 to 1982 for subcooled nucleate boiling in a circular pipe with heated wall under pressurized conditions.



The slide shows details of the testcase geometry and the investigated flow conditions.

Numerical Grids

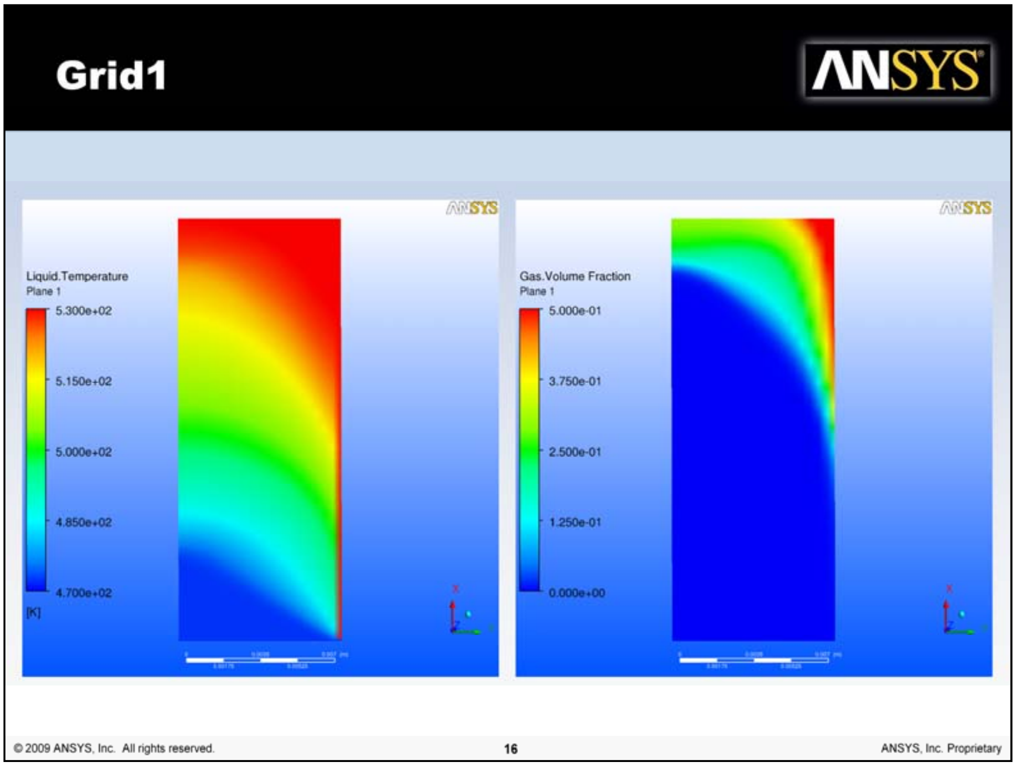
ANSYS

- **Validation on mesh hierarchy with regular refinement factor of 4 (2d meshes)**

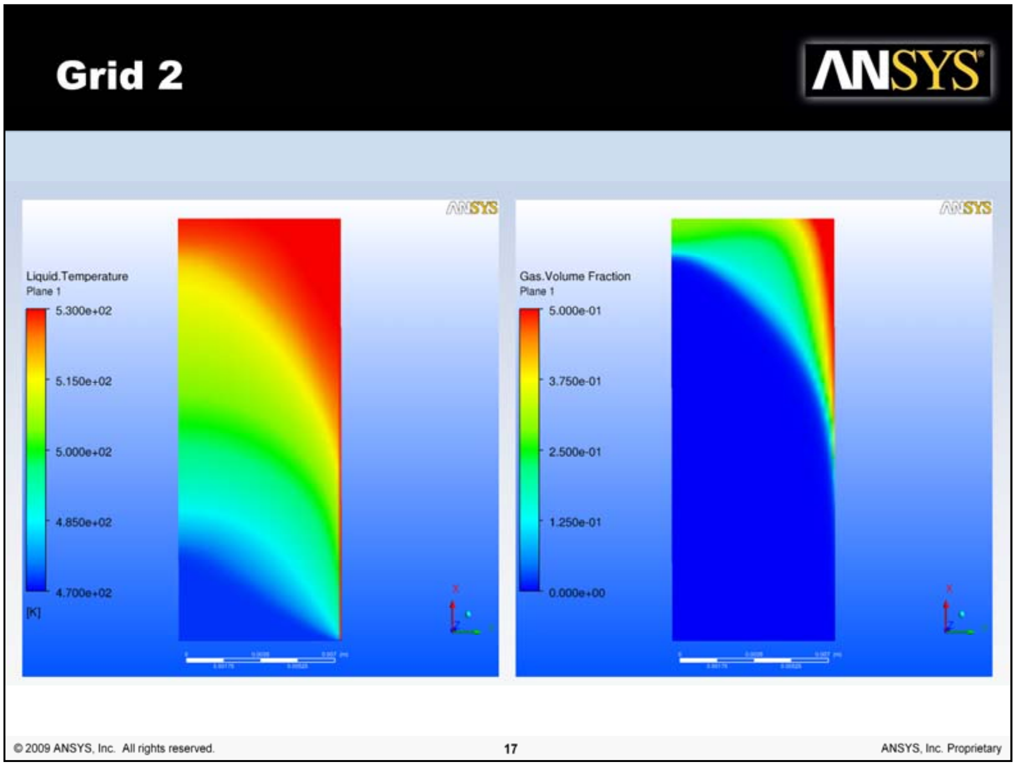
Grid	Grid1	Grid2	Grid3
# Nodes (uniform)	20x150	40x300	80x600
Max y^+	264	133	69
Δt [s]	10^{-2}	10^{-3}	$5x10^{-4}$

 15

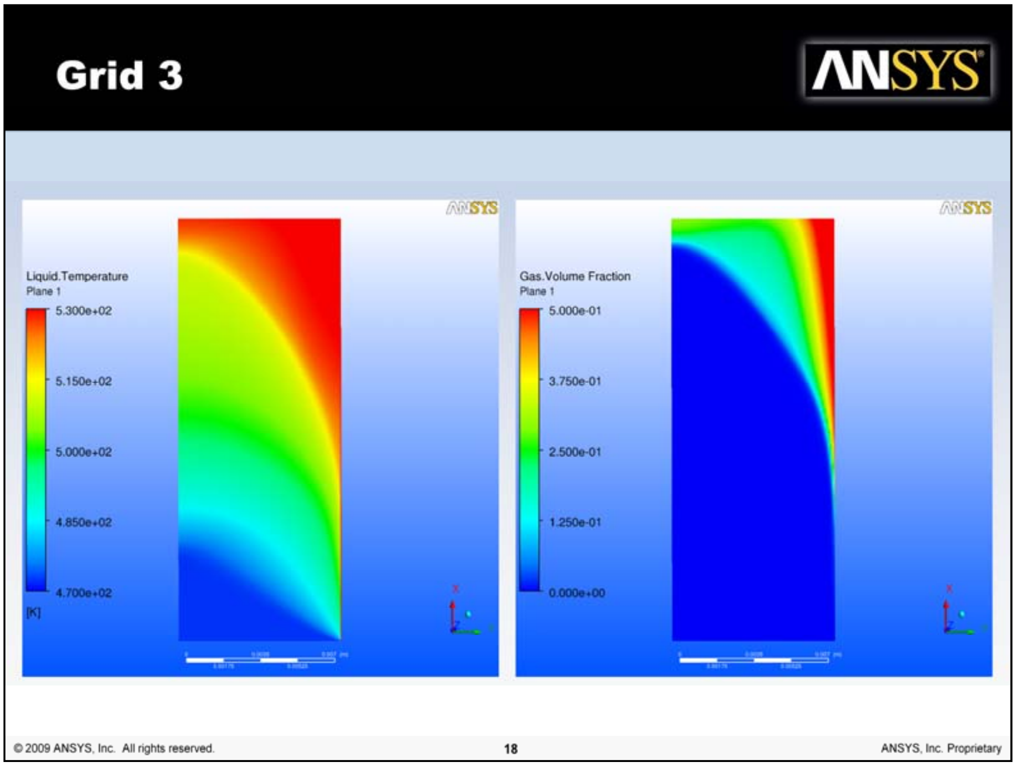
Investigation of nucleate subcooled boiling in the pipe of the Bartolomei testcase has been investigated on a hierarchy of 3 subsequently refined meshes with the above given properties. The timescales required for proper convergence of the steady-state simulations are listed as well.



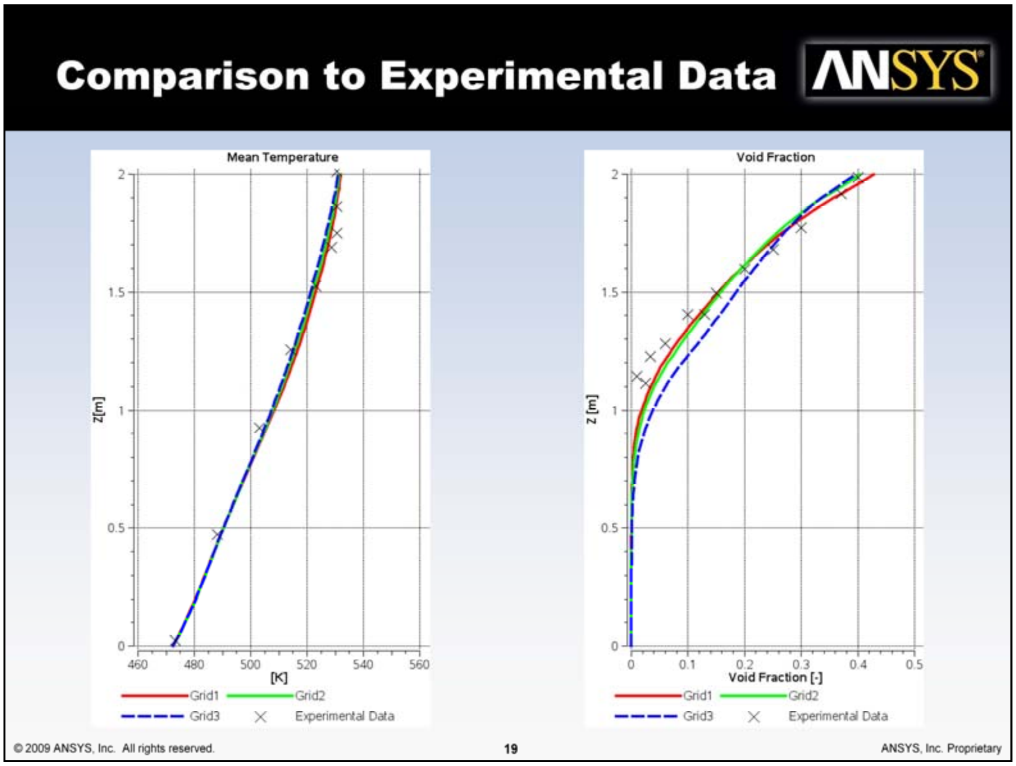
Distribution of liquid temperature and steam volume fraction on mesh level 1. No wall lubrication force has been applied in these cases.



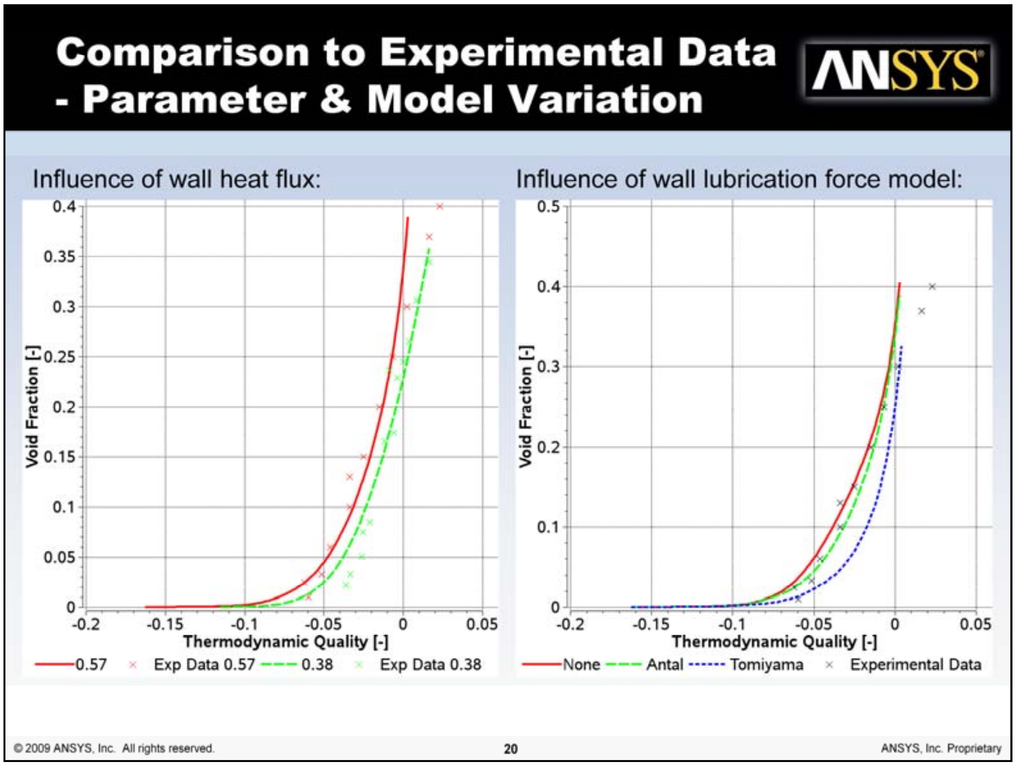
Distribution of liquid temperature and steam volume fraction on mesh level 2. No wall lubrication force has been applied in these cases.



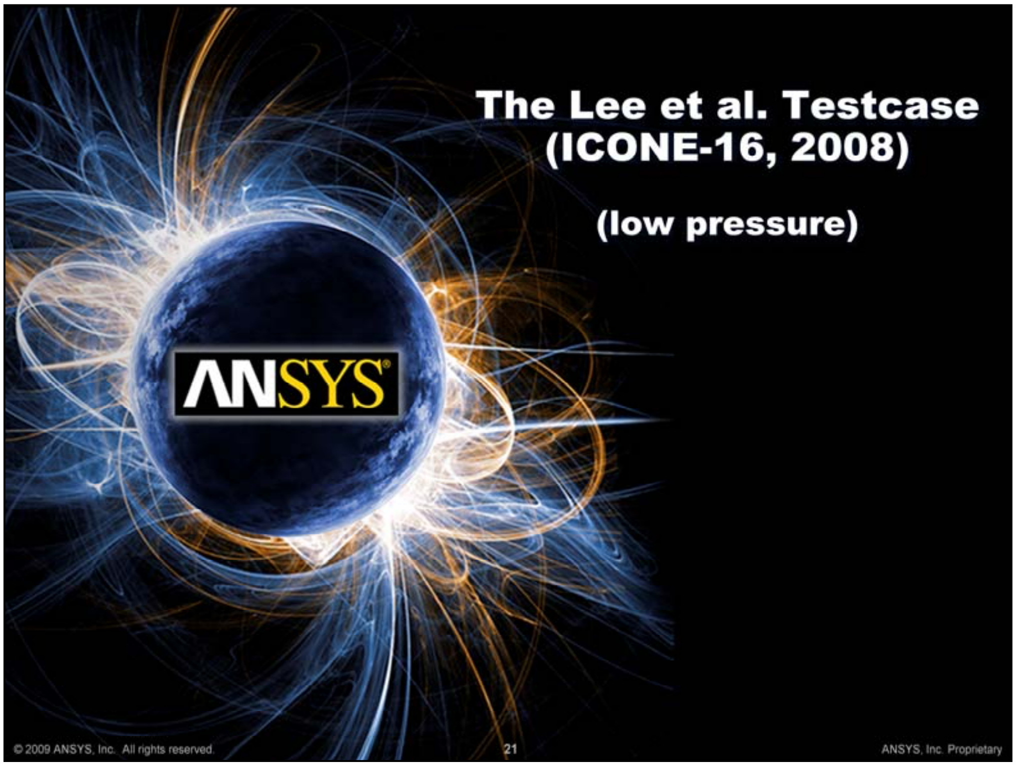
Distribution of liquid temperature and steam volume fraction on mesh level 3. No wall lubrication force has been applied in these cases.



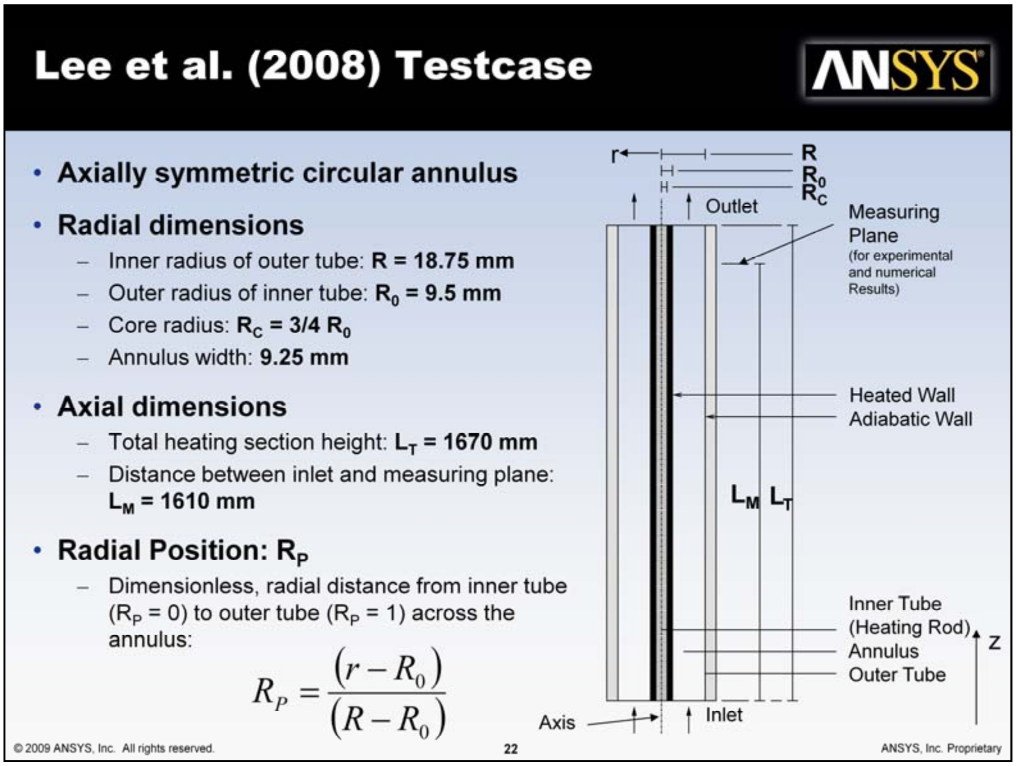
Grid dependency study for axial distribution of cross-sectional averaged liquid temperature and steam volume fraction. Diagrams show comparison to the experimental data of Bartolomei.



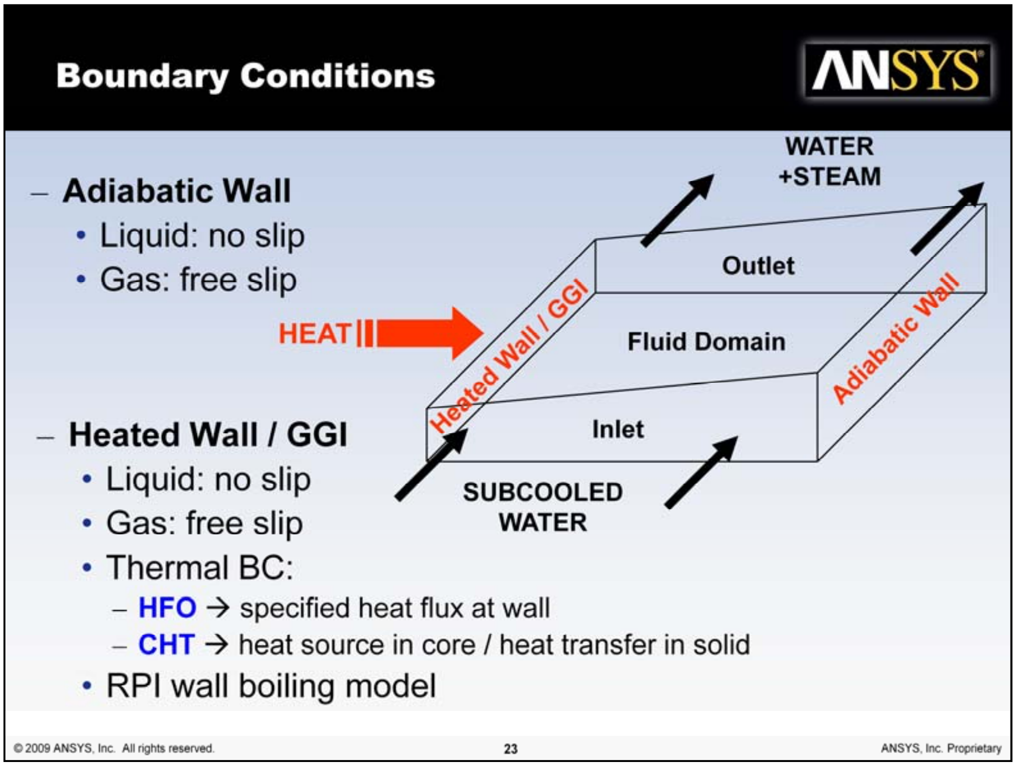
Parametric study with respect to applied wall heat flux and used wall lubrication force model formulation in the CFD setup. Diagrams show comparison to the experimental data of Bartolomei.



Wall boiling model validation using the testcase of Lee et al. (published on ICONE-16, 2008) for low pressure conditions.



Slide shows the details of testcase geometry of the Lee et al. testcase – boiling in a circular annulus heated by a heated rod on the symmetry axis. In similarity to a fuel rod in a fuel assembly of a nuclear reactor, the heater rod has been divided into a rod core, where all of the thermal energy is released and a cladding material, which is not actively heated and is only subject to heat conduction from the heated core to the fluid-solid interface. Dimensions of the flow geometry are provided.



Slide shows the applied boundary conditions. For the surface of the heater rod we have two different options:

a) prescribe a constant wall heat flux

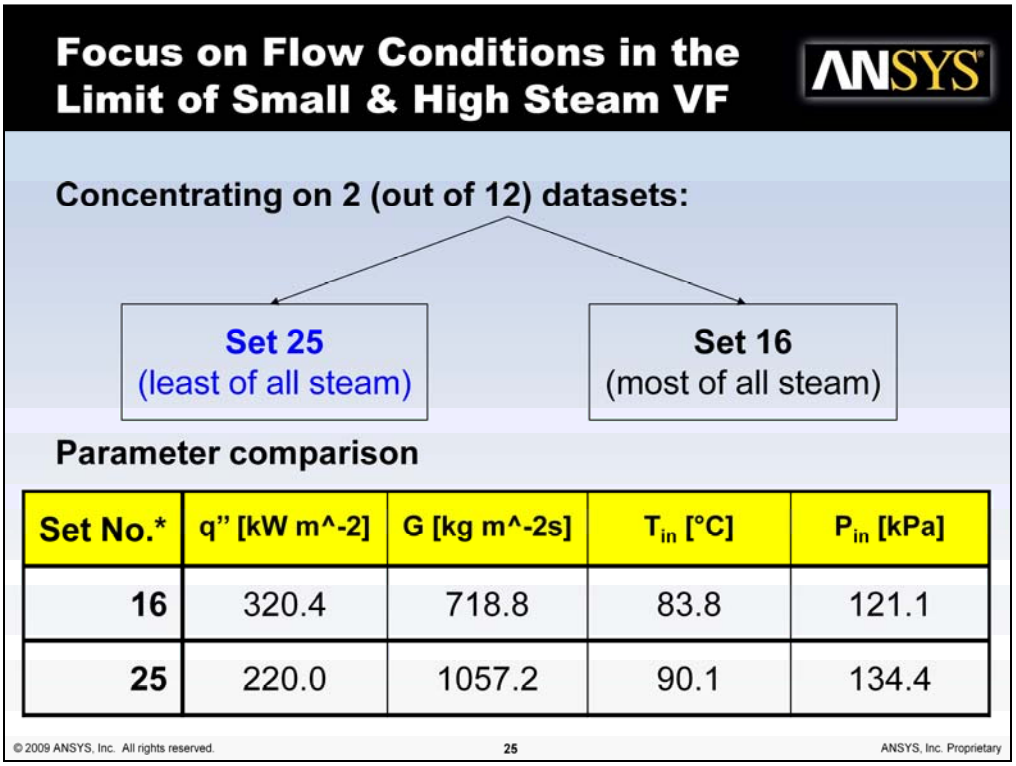
b) mesh the solid domain of the heater (core & cladding) and use CHT for prediction of the heat conduction; in this case the thermal energy is brought into the solid material by the prescription of a constant volumetric energy source in the rod core material

Mesh Hierarchy

Mesh Name		Grid 01 (coarse)	Grid 02 (medium)	Grid 03 (fine)
Domains (1 = HFO, 2 = CHT) *		1 | 2	1 | 2	1 | 2
No. of Nodes		1: 6342 2: 12684	1: 24682 2: 49364	1: 97362 2: 194724
No. of Elements (hexahedra)		1: **20x150** 2: **40x150**	1: **40x300** 2: **80x300**	1: **80x600** 2: **160x600**
y^+_{max} (at 1st node near wall)	Set16	~84	~41	~24
	Set25	~88	~45	~25
Tstep Δt [s]	Set16	0.001	0.002	0.0002
	Set25	0.1	0.0125	0.0002

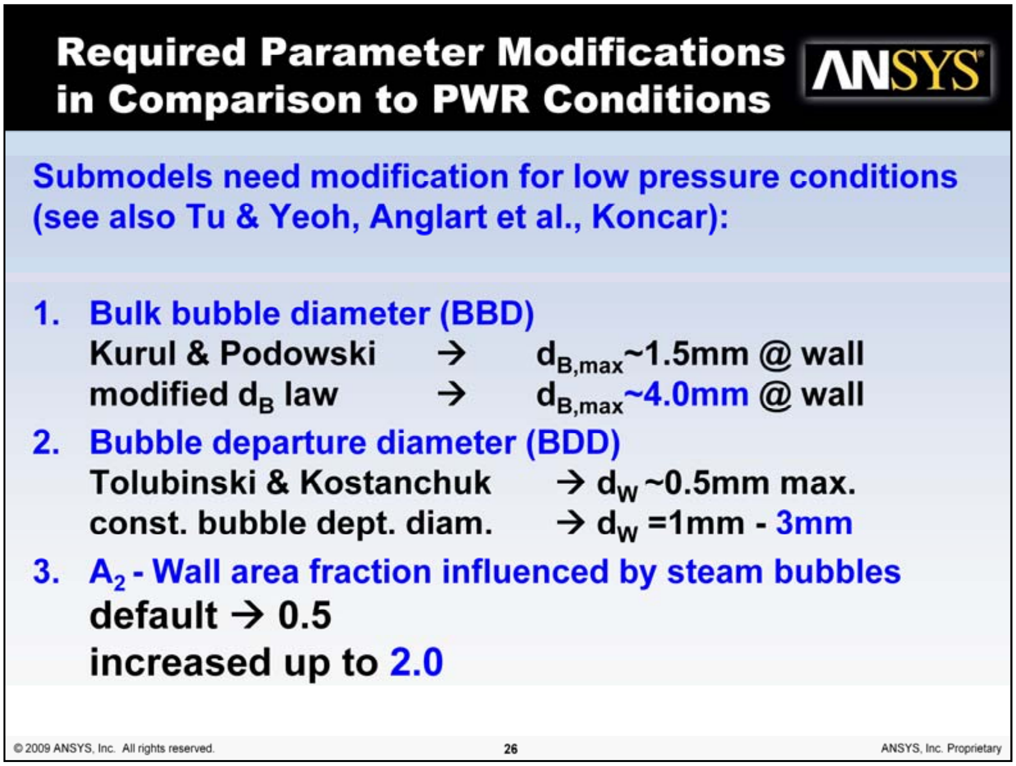
 24

Again, the investigation has been carried out on a hierarchy of subsequently refined meshes. Mesh parameters, near wall distance of the first mesh cell and corresponding integration time scales used for converged CFD solutions are given in the above table. Investigations have been carried out for two different sets of flow conditions – see next slide. For the Set25 both types of CFD setup (prescribed wall heat flux & defined volumetric energy source with CHT) has been investigated.



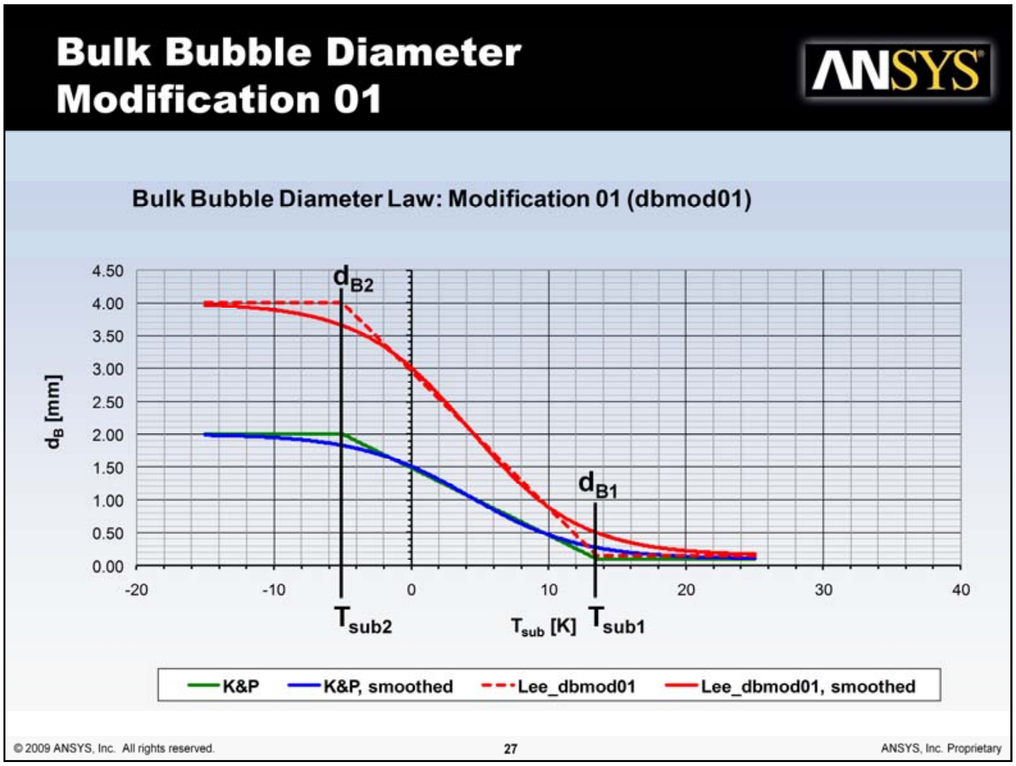
Set No.*	q" [kW m^-2]	G [kg m^-2s]	T_{in} [°C]	P_{in} [kPa]
16	320.4	718.8	83.8	121.1
25	220.0	1057.2	90.1	134.4

The paper of Lee et al. contains experimental data from 12 different experiments for varied flow conditions (pressure, massflow rate, inlet liquid temperature) and wall heat flux. The present investigation has focused on rather two datasets for the case, where the least of all steam has been produced and for the other limiting case, where the most of all steam occurs due to the wall boiling. In accordance with the paper of Lee et al. the datasets are named Set25 and Set16.

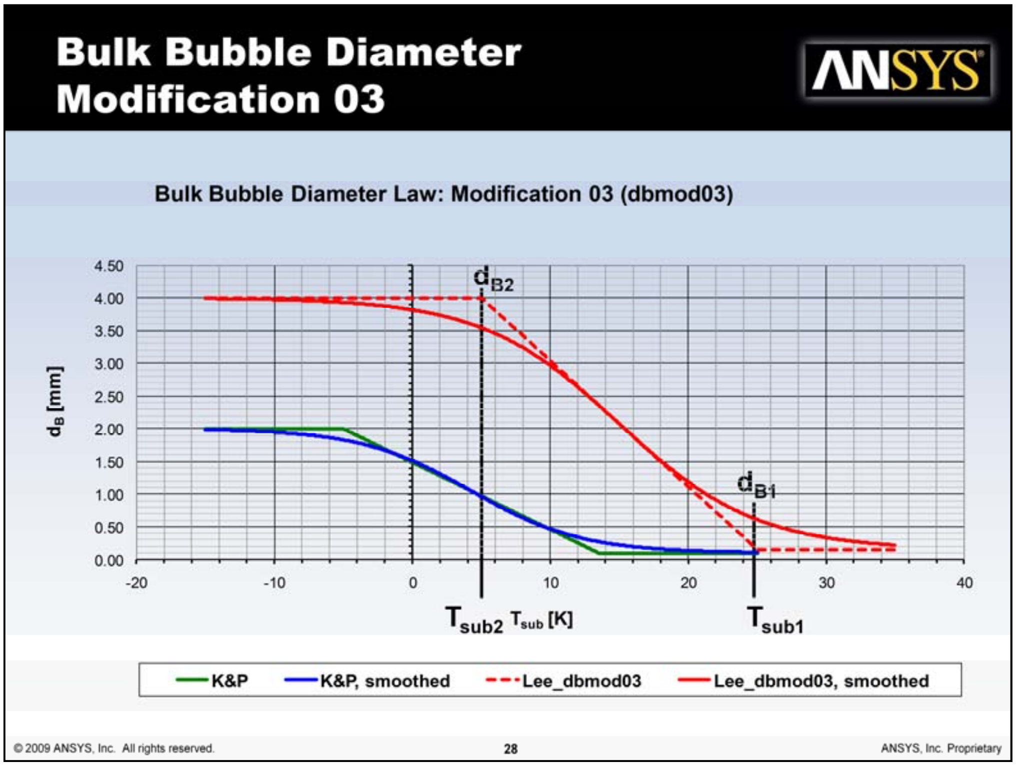


In comparison to the validation carried out for pressurized conditions of the Bartolomei testcase (or conditions in PWR nuclear reactors), for the conditions of the Lee et al. testcases some of the submodels of the wall boiling model need adjustment. This mostly applies to the prescribed law for the bulk bubble diameter, since the Kurul & Podowski correlation originally results in rather small bubble diamters of 1.5mm and less, while in the experiments of Lee et al. larger bubble sizes of 3-4mm have been measured.

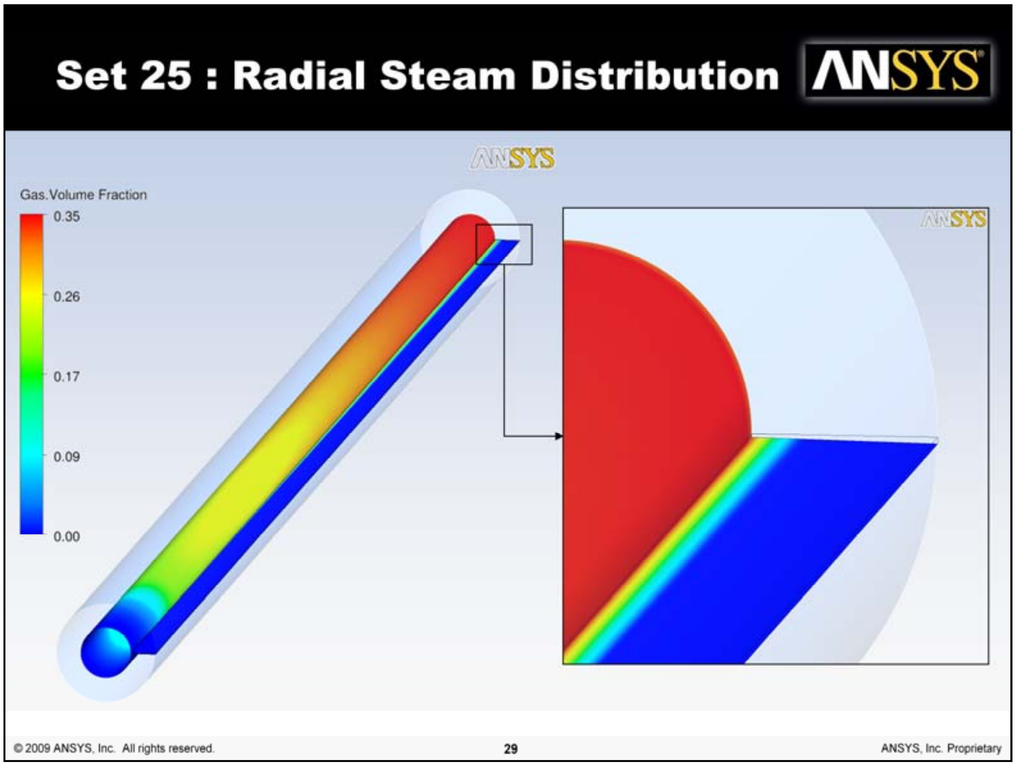
A further required modification applies to the bubble departure diameter for almost the same reason. Good results could be obtained for a bubble departure diameter of 3mm. And a last modification applies to the A_{2F} factor in the model formulation. It was observed that this factor with its default of a maximum of 0.5 was rather artificially limiting the evaporative heat flux and thereby the steam production from wall boiling. Increase of this model parameter has raised the limititation and has led to better agreement with data.



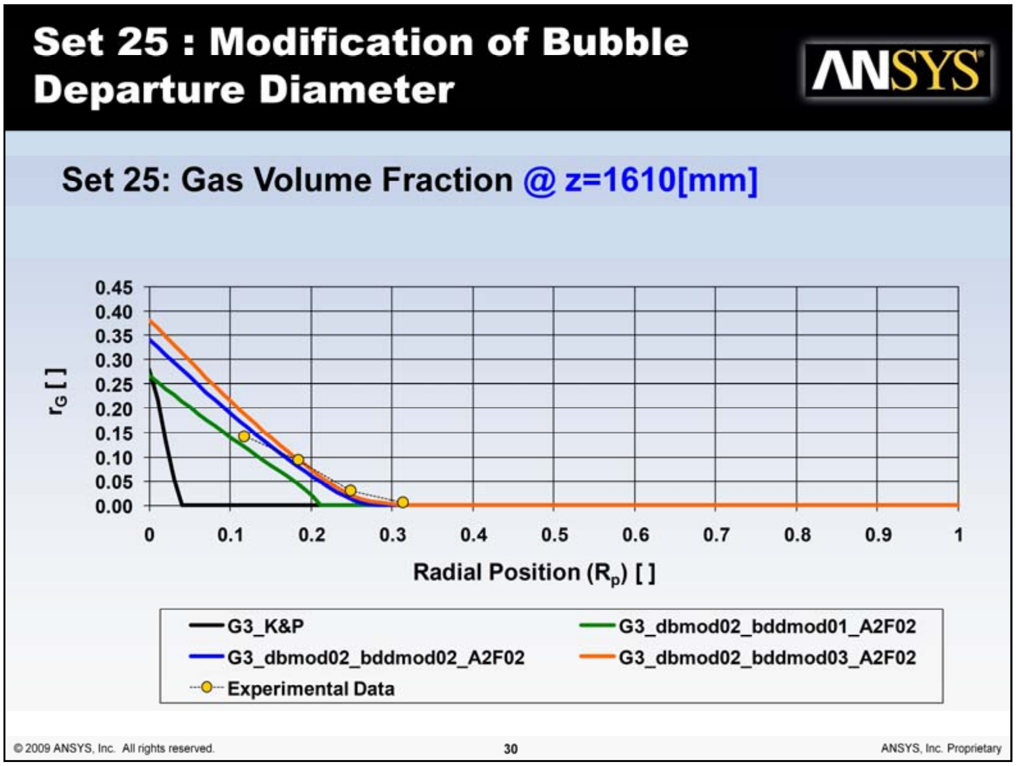
Slide shows the initial bulk bubble diameter distribution from the Kurul & Podowski correlation and the first modification undertaken for the present testcase conditions – increase of d_{b2}.



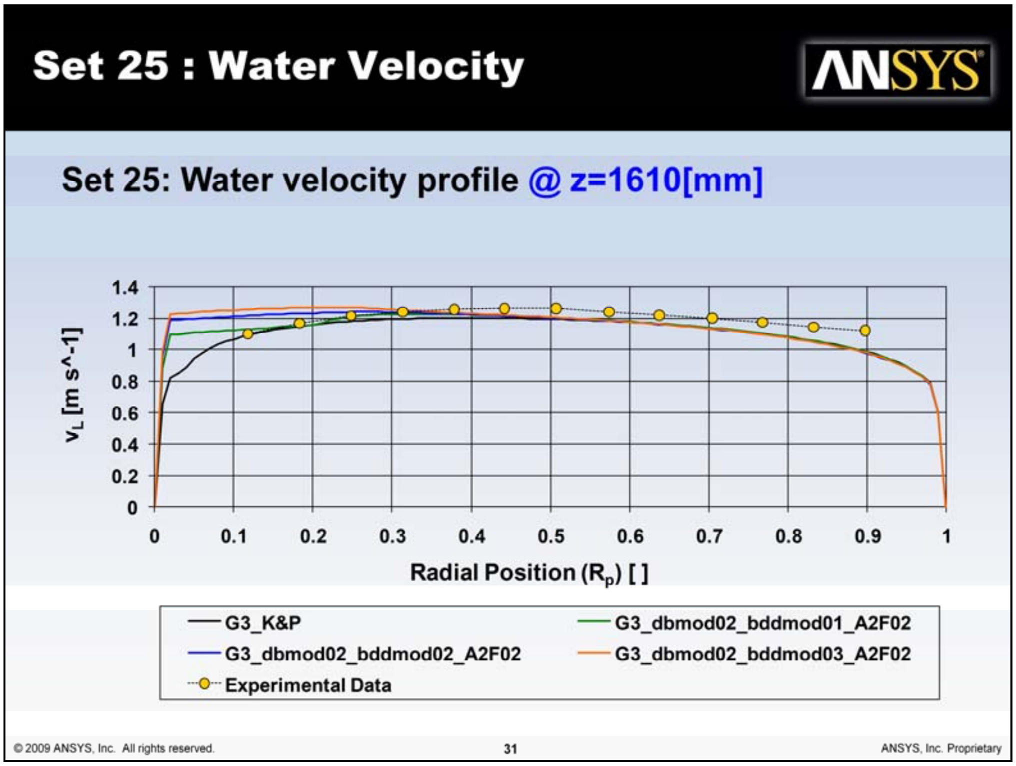
Slide shows the bulk bubble diameter distribution from the Kurul & Podowski correlation and the final modification undertaken for the present testcase conditions – shift of Tsub1 and Tsub2 towards larger liquid subcooling temperatures.



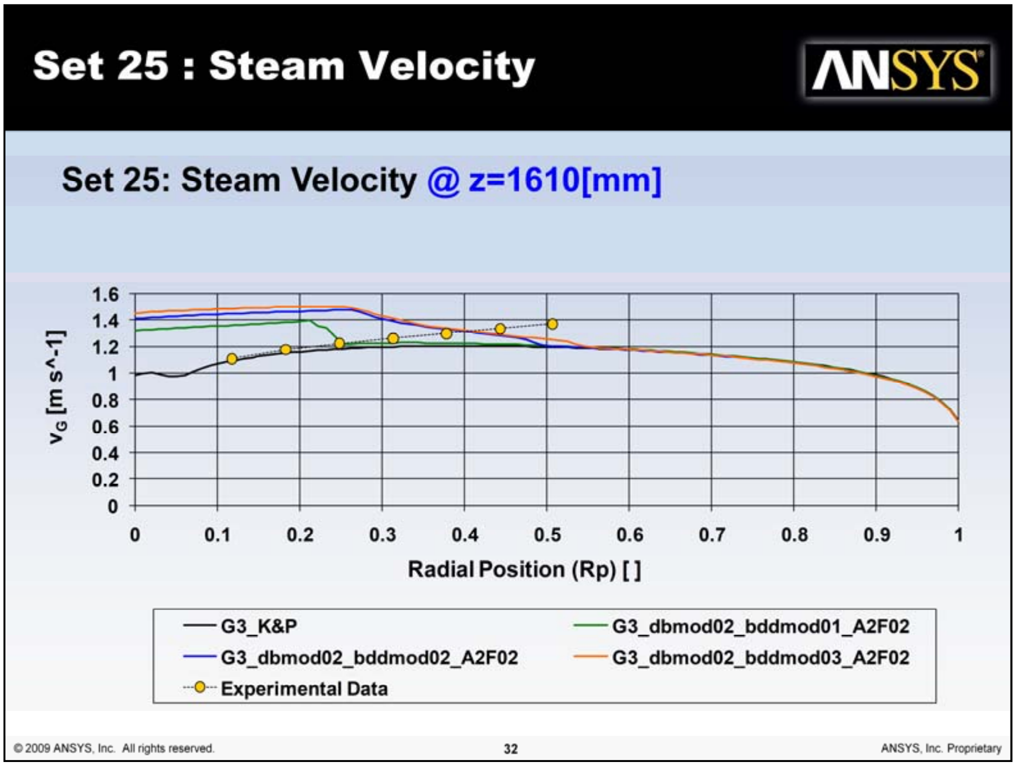
Radial steam distribution i the circular annulus of the Lee testcase configuration. Color on the rod of the heater represents the local steam volume fraction as well, reaching values of up to 35% of steam close to the outlet cross section.



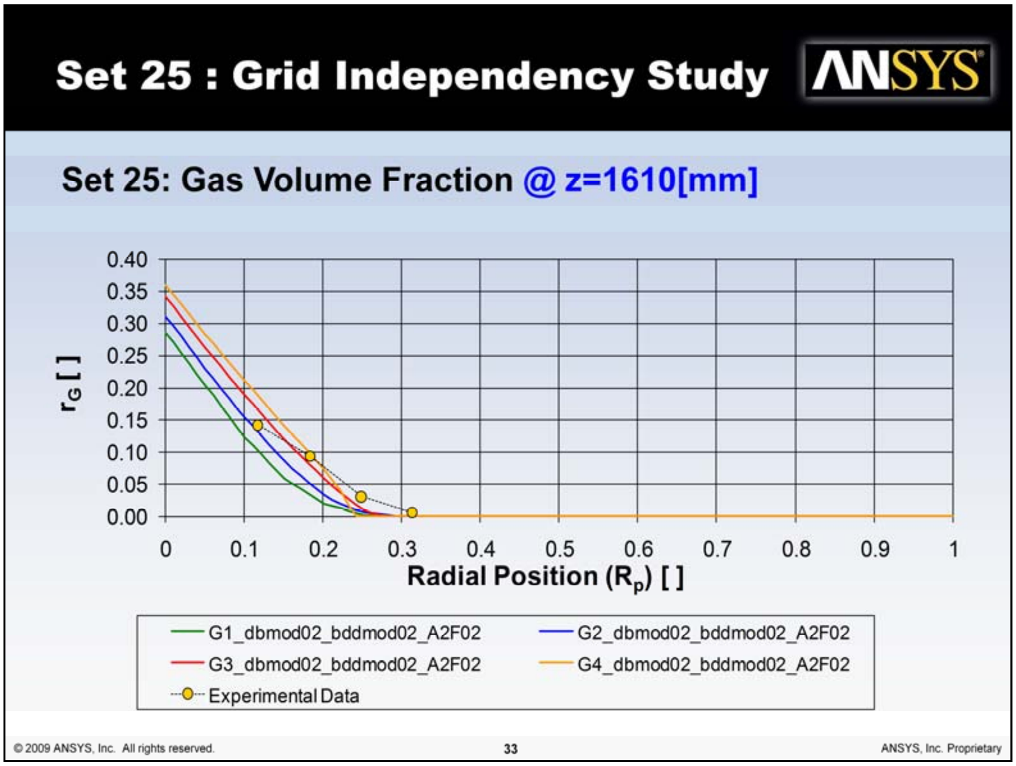
Cross-sectional steam volume fraction distribution close to the outlet cross section of the circular annulus in the Lee testcase. The diagrame shows the influence of the different settings for the bubble departure diameter. The black curve show the very initial result with the use of the Tolubinski & Kostanchuk and the Kurul & Podowski correlations, substantially underpredicting the steam production in this case. The final result is in good agreement with the experimental data of Lee et al..



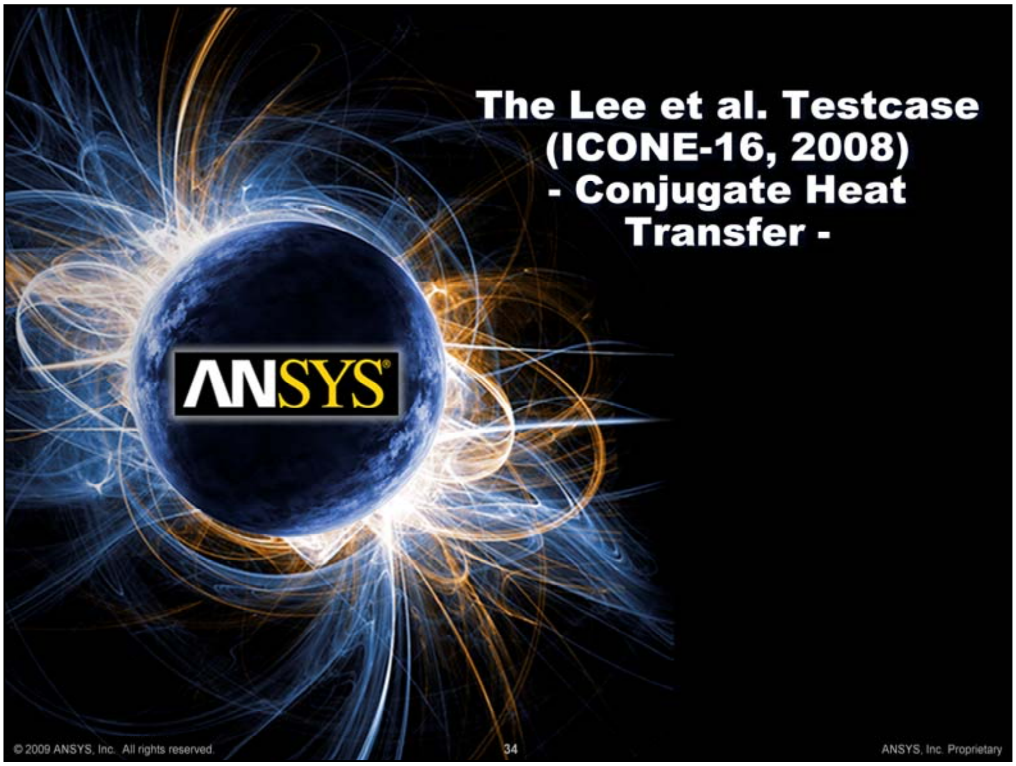
Cross-sectional distribution of the water velocity close to the outlet cross section of the circular annulus in the Lee testcase. The CFD result overpredicts the water velocities on the heated surface due to the prescribed free-slip boundary condition for the steam phase and the resulting strong influence/acceleration from bubble buoyancy. In reality the developing steam bubbles grow on the surface until they begin sliding motion along the heater surface. Therefore a free-slip boundary condition is not appropriate and has to be modified in future CFD simulations.



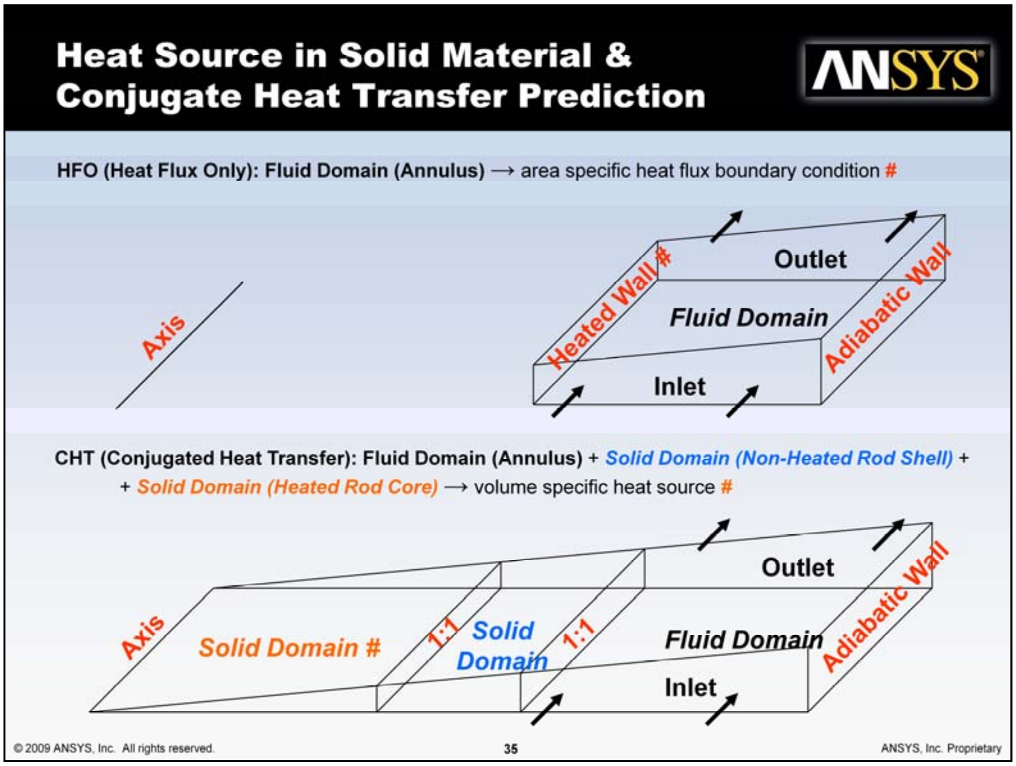
Cross-sectional distribution of the steam velocity close to the outlet cross section of the circular annulus in the Lee testcase. Experiments show much smaller steam velocities close to the heated surface then in the CFD simulation, which seems to be rather in a contradiction with the measured bubble sizes and the buoyancy effects which should result out of this. As explained the free-slip BC for the steam phase is not fully satisfied for the CFD, but it seems not justified as we4ll, that the fluid phase should see almost no effect from the massive steam production at the heater wall as in the previous diagram. Experimental data seem not to be consistent in this regard and possible measurement errors are not commented in the paper of Lee et al.



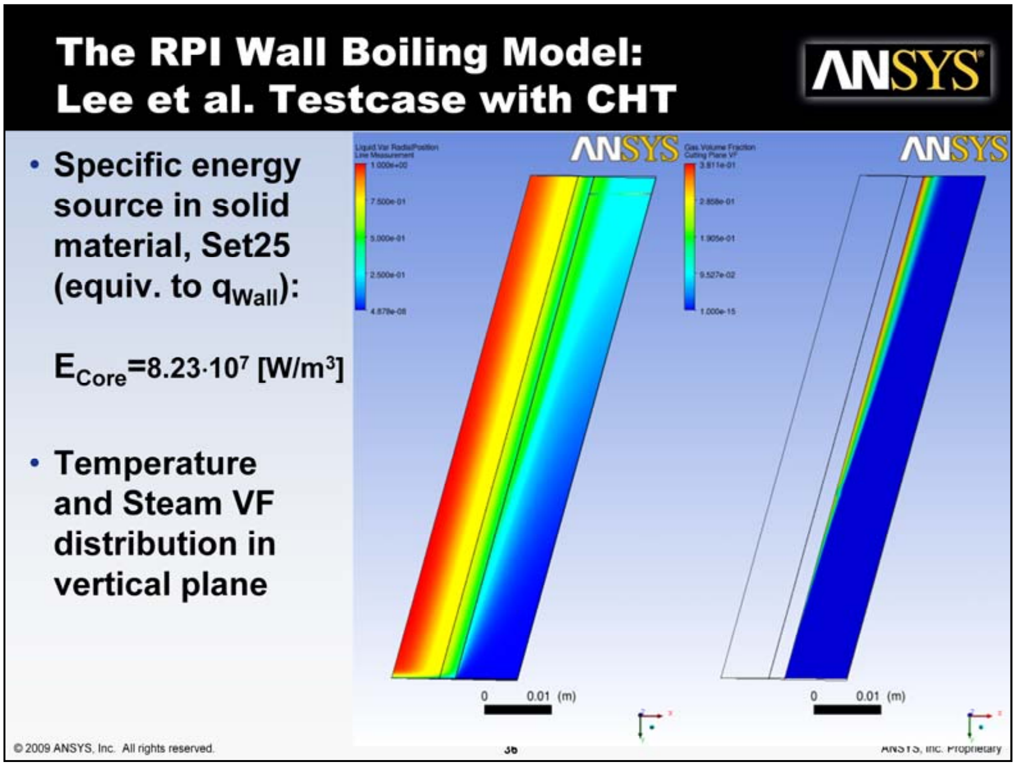
Grid dependency study for the finally established set of model parameters for Set25 flow conditions. The CFD result is not yet fully grid independent, but on meshes 3 and 4 in rather good agreement with measured steam volume fraction data.



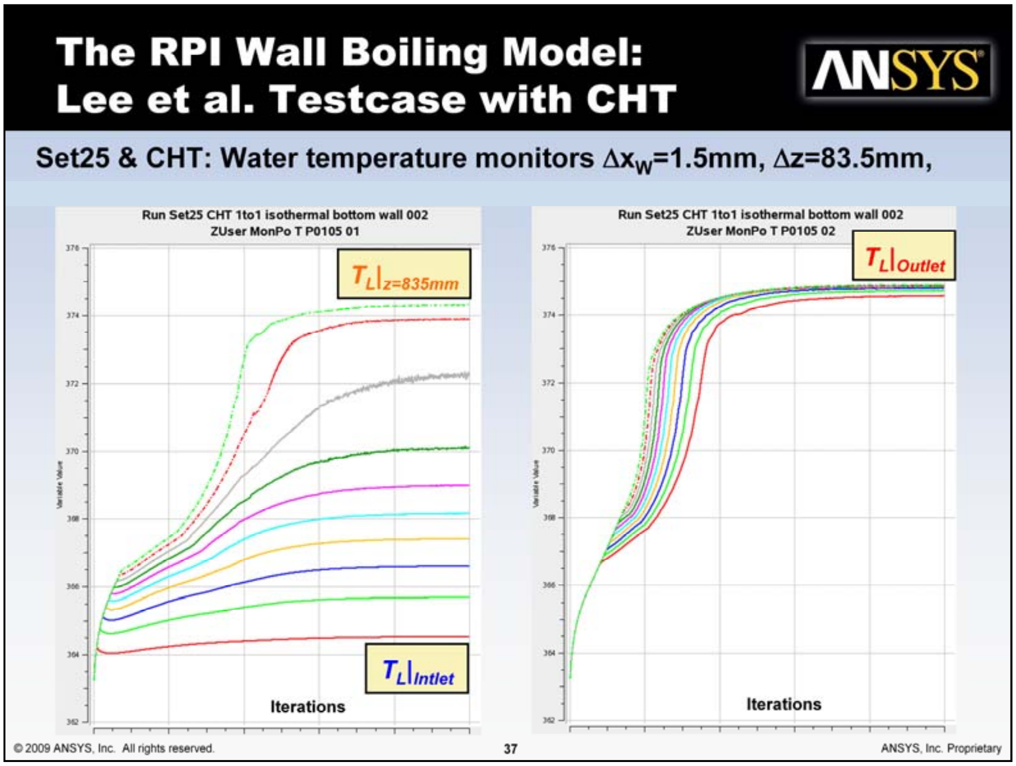
Combination of the modified RPI wall boiling model with CHT in the solid material for the Set25 of the series of Lee et al. testcases.



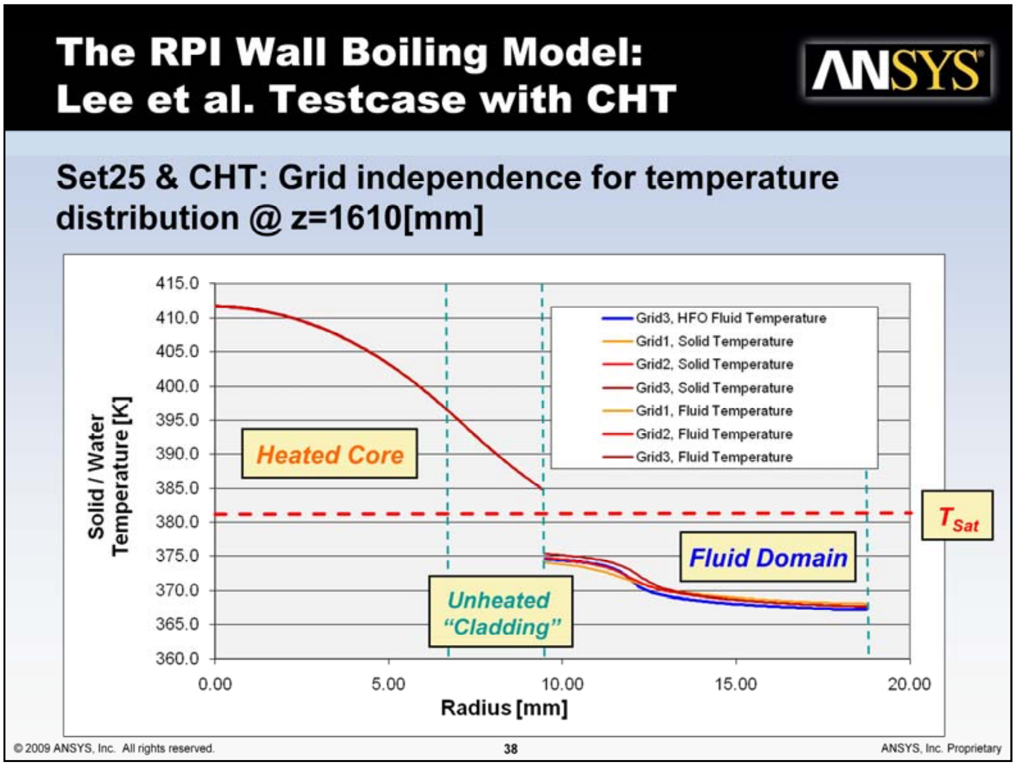
From the prescription of a constant wall heat flux at the heated surface we switch to a model setup, where the thermal energy is equivalently introduced to the computation through a constant volumetric energy source in the core material of the heated rod, thereby resolving the conjugate heat transfer (CHT) in the solid domain by solving an energy transport / heat conduction equation in the 2 solid domains. In this case no specific thermal boundary condition is required for the specified fluid-solid interface. In this particular simulation the mesh resolution on both sides of the fluid-solid interface was identical (1:1 interface).



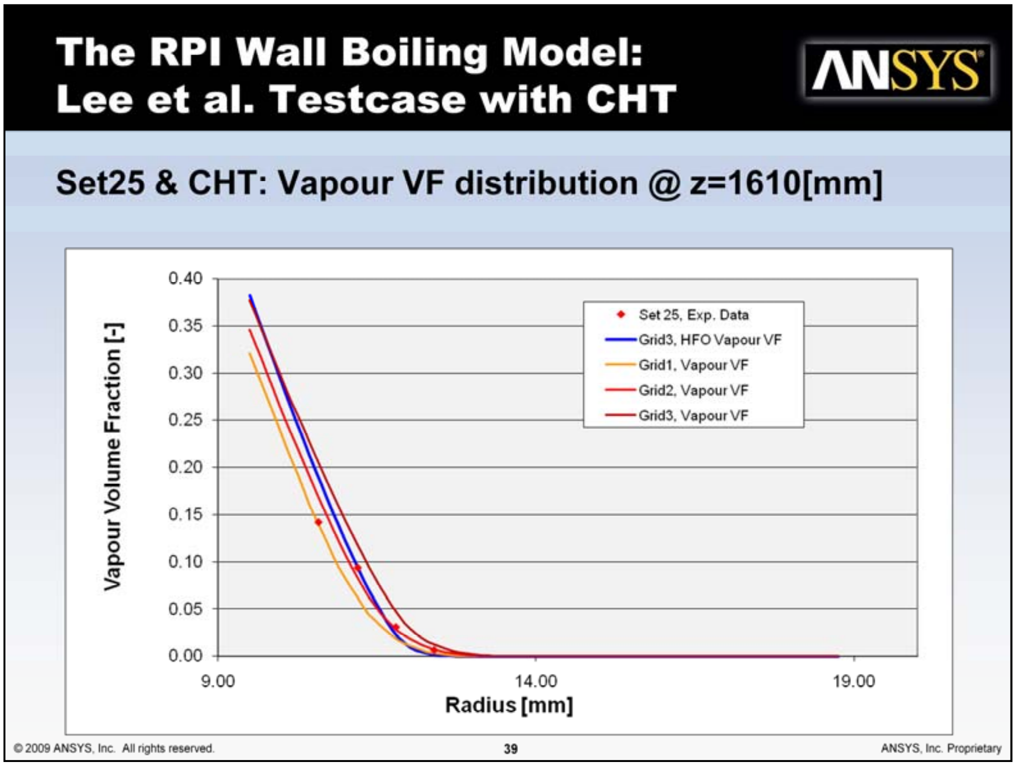
Planar distribution of temperature (solid and liquid) as well as steam volume fraction in a vertical cross section of the testcase configuration. It can be observed how the boiling develops with increased height in the circular annulus and how the liquid phase gets gradually heated up by convective heat transfer and steam recondensation in the subcooled liquid.



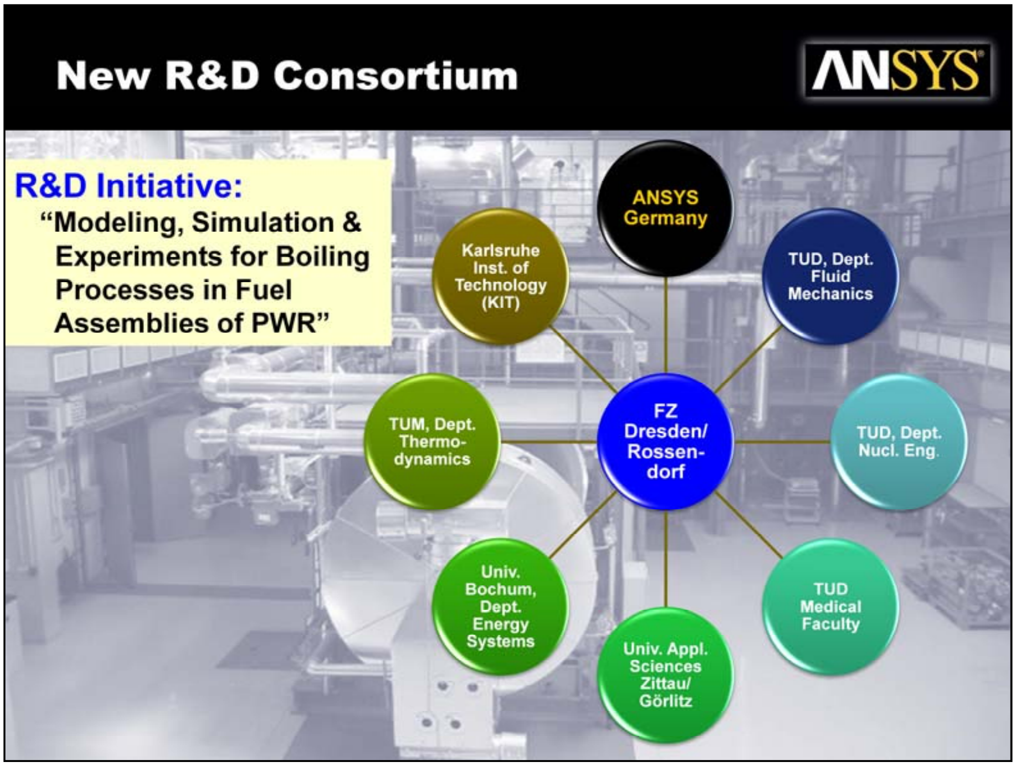
Development of liquid temperatures in a row of monitoring point locations close to the heated surface from bottom to top of the circular annulus fluid domain. After certain number of iterations the predicted liquid temperatures arrive at steady state.



Cross-sectional temperature distribution close to the outlet cross section both in the solid and fluid domain. It can be clearly observed, that the temperature in the core material of the heater is by far not a constant. Furthermore a step function in temperature can be observed directly on the heater surface with a temperature difference of about 10K between the cladding material and the liquid temperatures. The CFD results from the last simulations are fairly grid independent and agree well with the CFD simulations for the prescribed wall heat flux.

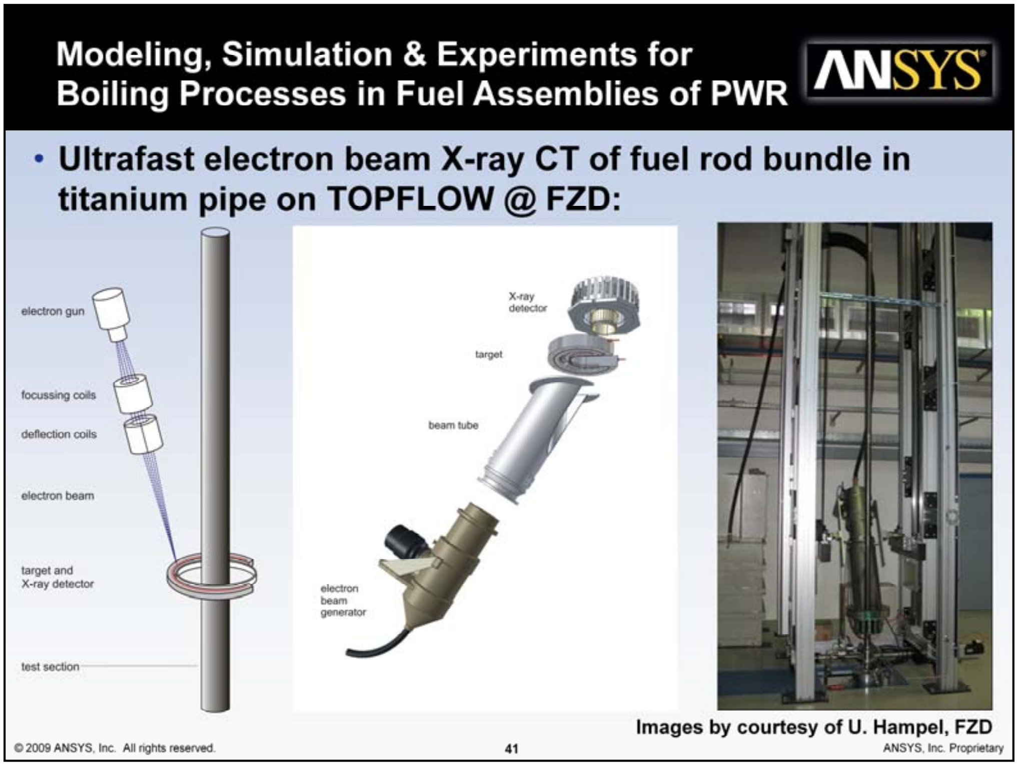


Grid dependency study for the finally established set of model parameters for Set25 flow conditions and for the combination of CHT and wall boiling model. The CFD result is again not yet fully grid independent. The agreement with the prescribed wall heat flux simulation and with experimental data is nevertheless rather good. The result from the CHT simulation on mesh 3 shows slightly higher steam volume fraction then previously on mesh 3 with prescribed constant wall heat flux.

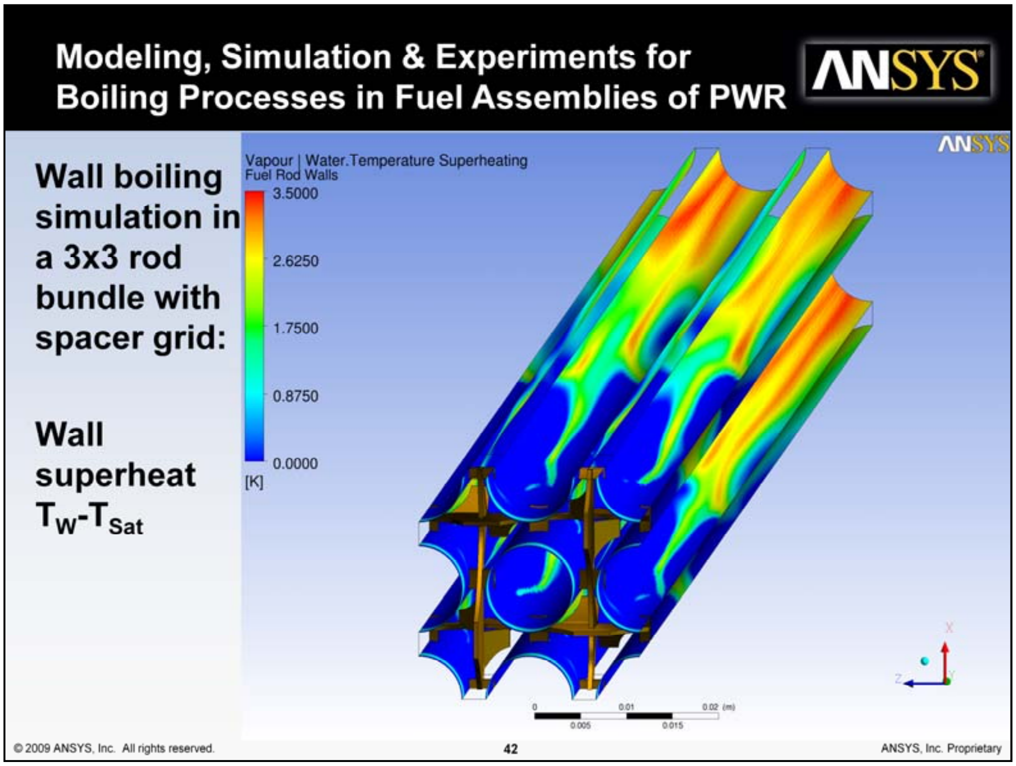


ANSYS will continue its efforts in R&D for development of multiphase flow models and of wall boiling model in particular. Therefore ANSYS has joint a large R&D consortium and program, which will focus for the next 3 years on modeling, simulation and experiments for boiling processes in fuel assemblies in PWR nuclear reactors. This research will be sponsored by the Federal German Ministry of Education & Research (BMBF). ANSYS is hereby engaging in tight collaboration with leading German universities and research centers in the further development of CFD models for complex flow phenomena in multiphase flows including flows with strong heat and mass transfer like in NRS and nuclear engineering applications.

Former R&D programs with participation of ANSYS has been sponsored by the German Federal Ministry of Economy (BMWi) and the R&D grant to ANSYS is hereby gracefully acknowledged.



The R&D collaboration offers the unique opportunity to validate CFD models based on detailed local and non-intrusive measurements for boiling flows under pressurized conditions. The main validation experiment will be carried out by FZ Dresden-Rossendorf using X-ray computer tomography for the measurement of local flow parameters in a heated rod bundle enclosed in a vertical titanium pipe.

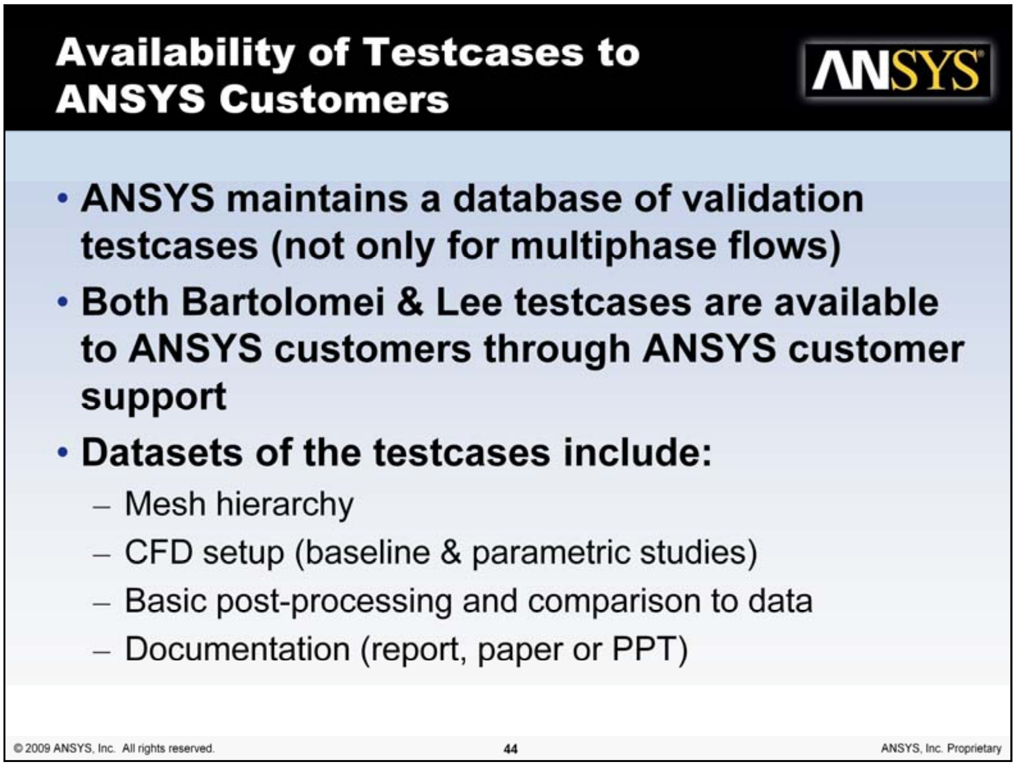


The experimental work will be aligned with further mathematical-physical model development and flow simulations for boiling flows in fuel assemblies of PWR using ANSYS CFD.

Summary & Outlook

ANSYS

- ANSYS CFD 12.0 provides set of CFD models for simulation of boiling processes
  → subcooled nucleate boiling model
- Wall boiling model validation has shown good agreement with experimental data
- Identification of submodel parameters for high & low pressure conditions of subcooled boiling
- Modified RPI wall boiling → almost grid independent
- Model compatibility with CHT (Conjugate Heat Transfer) in solid material
  → use CFX-12.1 for GGI interfaces (1÷1 works in CFX-12.0)
  → still too small time scales required for convergence
- ANSYS undertakes further R&D for model improvement



Reference to the ANSYS model validation testcase library – if you are interested in this material file your inquiry through your ANSYS customer support.

Thank You!