MODELING OF THE DYNAMICAL BEHAVIOR OF POWER PLANTS WITH A COMMERCIAL SOFTWARE SHELL

Wolfgang Zehtner[1], Prof. Dr.-Ing. Hartmut Spliethoff[1], Dr.-Ing. Wolfgang Woyke[2]

[1]TU München, Lehrstuhl für Energiesysteme
Boltzmannstraße 15, 85748 Garching bei München, Germany
[2]E.ON Energie AG
Brienner Straße 40, 80333 München, Germany
zehtner@es.mw.tum.de (Wolfgang Zehtner)

Abstract

The demands on modern power plants increase continuously. Highest efficiency is to be combined with high flexibility for load changes. Due to these constraints manufacturers are faced opposite demands in the layout of the power plants. Utilities are faced a similar discrepancy.
To find an optimal solution, numerical simulation of steady-state and dynamical behavior of power plants has become indispensable. Modern simulation platforms no longer depend on programming skills but enable any engineer to use them after short training courses. Most programs for the simulation of plant dynamics use one-dimensional discretization, so the plant geometry can not be modeled in detail. Resulting inaccuracies can be avoided by adjusting the discretization scheme and default model parameters manually.
In the following paper, at first the basic principles of simulation of steady-state and dynamics are presented. In the second chapter, the discretization scheme of APROS, the used system shell is introduced. After that, common plant components are discussed that differ significantly from the draft approach proposed by the simulation software. Finned pipes and finned tubes are presented in detail. Approaches to deal with the special characteristics of the simulation software and the validation comparing to operation of a coal fired power plant end the report.

Keywords: coal, power plant, modeling techniques, validation.

Presenting Author's Biography

Wolfgang Zehtner works as a PhD student at the Institute of Energy Systems at the TU München. He completed his degree on Electrical Engineering at the TU München in spring 2004. His studies were already focused on power engineering. Currently, his doctoral studies are concentrated on improving startup and load change phases of coal-fired power plants. As his research work contains the behavior of lots of different power plant components, a sophisticated simulation program package is used.

ISBN 978-3-901608-32-2

1 Introduction

1.1 History

In the last decades, numerical simulation has become an important part in engineering practice and especially in power engineering. Lots of different physical effects take place in power plants and have to be modeled. Computer technology and user friendly software shells enable an accurate and efficient modeling of power plants. The simulation software calculates the steady-state behavior of power plants based on detailed information of heat transfer and conduction, fluid properties, combustion reactions and wall friction. The conservation laws of mass and energy are fulfilled. These programs have become essential tools for the design of most industrial appliances, especially for such large ones as large scale coal fired power plants.
For the simulation of dynamics more advanced software is necessary, as more physical effects must be taken into account. Storage and inertia effects, heat capacities and fluid storage must additionally be included as well as the conservation law for the momentum. The user has to provide additional and more detailed information about the object of the simulation.
Due to the increased complexity, the demands on computational hardware are also significantly increased. For these reasons, numerical simulation of plant dynamics has become relevant during the last years. Few very special appliances, as the safety research in the nuclear field, were working in this field. In Germany, the Gesellschaft für Anlagen- und Reaktorsicherheit (GRS, nuclear safety research institute) [1] has developed the thermal-hydraulic computer code ATHLET (Analysis of THermal-hydraulics of LEaks and Transients) [2] for the analysis of transients, leaks and large breaks in light water reactors [3]. VTT Technical Research Centre of Finland [4] also performed research in this field and developed APROS (Advanced Process Simulator) [5] to achieve very much the same purpose [6]. Due to high costs for the hardware and the lack of a graphical user interface (GUI) in former times, the users of the programs came mainly from the scientific side and commercial usage of such kind of programs was unthinkable yet.

1.2 Actual situation

The development of more powerful computers gave a boost to the development of simulation tools. Software companies like The MathWorks were founded and developed software for technical computing that was easier to use as it contained a GUI or a simplified programming language developed specially for mathematical problems [7]. Commercial distribution of numerical software became possible. Programs like these spread quickly in universities and in research and development facilities in the industry.
VTT improved APROS, the program shell used in this application. The system now provides a graphical user interface (GUI) that visualizes the model and offers a lot of predefined component models can be selected out of libraries. This opened the way for commercial distribution of the software platform.
Nowadays, also engineers without special programming skills are able to perform static and dynamic simulations of complex plants after short training courses [8].

1.3 Modeling paradigms

As numerical simulation systems have diversified, they are classified into several categories [8] [9]. For the simulation of processes from the user's point of view, the following classes are most important:

- **steady-state vs. dynamic modeling**
  In steady-state modeling the model provides data of certain, fixed operating points. Transients are only regarded in dynamic modeling tools such as APROS, the system shell used here.
- **distributed vs. lumped parameters approach**
  In the distributed parameter approach a considered fluid flow is subdivided in a finite number of spatial distributed control volumes. The local values of the parameters are calculated for each volume (e.g. $\rho(x, y, z)$). The result is the spatial distribution of the flow variables.
  In the simulation of processes in most cases the lumped parameter approach is used, since the received models are less complex and describe the reality with sufficient accuracy. This is also the case in APROS. For this purpose all of the process components that contain fluids are subdivided into a couple of finite volumes, which commonly match the geometric dimensions of the system components. The thermodynamical properties of the fluids are assumed to have the same value within the whole volume as it is common for a lumped parameter model (e.g. $\overline{\rho}$) [10]. By applying the conservation equations and adequate constitutive equations on the finite volume a system of differential equations in time is obtained and can be solved.

1.4 General survey about APROS

Software shells for simulating the dynamical behavior of large appliances are faced two main challenges: Demand for accuracy and demand for speed. Accurate calculations of heat transfer coefficients and temperature distribution in complex geometries with turbulent flow requires three-dimensional discretization as it is done in CFD-programs. These models consist of very much finite volumes which accordingly increases the hardware demand. For this reason, simulation of plant dynamics with three-dimensional models is only possible on high-performance data centers.
The demand for speed and for operativeness on an common office computer still prohibits these three-dimensional models. So one-dimensional models bridge the gap. Their decreased accuracy can successfully be compensated by validations, tuning factors and intelligent discretization, as described in detail in this paper. APROS, the system shell used here, consists of individual modules that are combined to build the

ISBN 978-3-901608-32-2

component models. The way of combination can extensively be modified by the user.
For most of the components used in power stations, APROS offers a library of roughly predefined models, that contain the correct equations and can be parametrized giving the geometry data. The component models in APROS are configured to model e.g. one straight pipe with cylindrical shape. This works good for most of the pipes in a power plant. Heat transfer is calculated by additional modules for heat transfer on the inner and outer wall surfaces and for the heat capacity and conduction in the wall that use heat transport equations. Up to three different layers in the wall can be defined that may consist of different materials. So, the heat distribution of a copper pipe insulated with stone wool and surrounded by an steel shroud can be modeled using the GUI.
As for cartesian, cylindrical and spherical coordinates, there are well-established formulas available, this approach gives good results for the behavior of straight pipes.
In power plants, there are also pipes whose properties and environment are not covered by well-established formulas. Their outer surface may not be circular or the heat transfer on the outer side may occur on different radii. So this simplified component model of one straight pipe cannot cover the very diverse properties of all real plant components. There are plant components, whose physical environment is defined that poor that there are no ways to exactly determine their behavior (e.g. fouling in heat exchangers). In this case the quality of numerical simulations is rather constraint.
For the user, detailed knowledge of the underlying discretization scheme is fundamental so that the component models can be chosen and adjusted for best modeling the reality. So a short overview on the discretization scheme is given in the following chapter. Knowledge of the plant to be simulated is as well important so that the importance of one specific plant component on the overall accuracy can be estimated reliably. Some examples of modeling plant components with very complex geometries are described in the following.

2 APROS - basic principles

2.1 Thermohydraulical discretization

In APROS, the finite volumes that contain fluids are represented by so-called *nodes*. The flow junction between two *nodes* is called a *branch*. Fig. 1 shows the symbols of two pipes, connected by so-called points and their component models below. In this example,

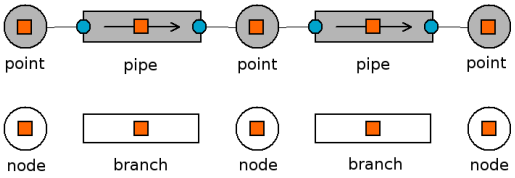

Fig. 1 simple thermohydraulical discretization

a correspondence of points to *nodes* and pipes to *branches* is clearly visible. On the calculation level, the *nodes* get assigned a certain part of the volumes of the adjacent *branches*. These are the finite volumes for applying the conservation equations for mass and energy. The volume and further geometry data of the branches (lenght, diameter, wall friction, etc.) are used for the conservation equation of the momentum. This creates a staggered grid of finite volumes which improves numerical convergence. [11]
To improve calculation accuracy in long pipes or

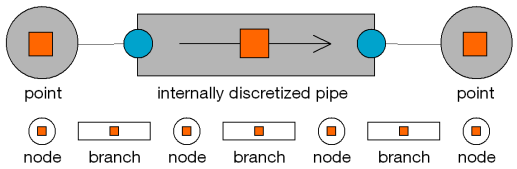

Fig. 2 more detailed thermohydraulical discretization

in heat exchangers, a more detailed discretization is possible as Fig. 2 demonstrates. In this case, the pipe is internally discretized in two extra nodes that are located between the outer points, in equal distance. The number of branches is increased automatically in respect to the number of nodes.

2.2 Discretization of solids

For the calculation of heat distribution in solid structures a similar discretization as a grid of *heat structure nodes* and *heat structure branches* is used that divides tube walls in radial direction. A *pipe with heat structure* is discretized as the left box in fig. 3 shows: A thermohydraulical *node* at the inflow of the *pipe with heat structure*, whose heat transfer to the wall is modeled by a *heat transfer module* is connected to the adjacent *heat structure node*. *Heat structure branches* connect further *heat structure nodes* in radial direction. Their number can be chosen freely according to the extent of discretization.
This pipe can be extended to a heat exchanger by

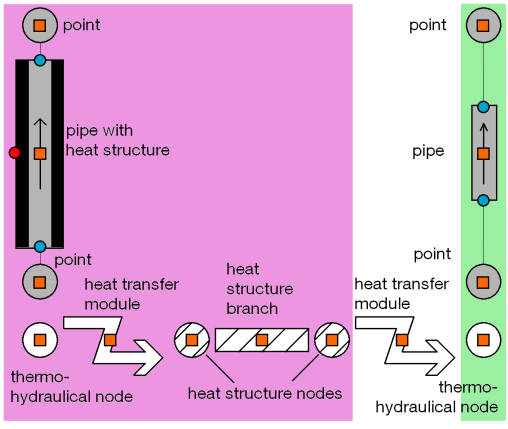

Fig. 3 simple discretization in heat exchanger

simply adding a pipe for the second fluid (right box) and connecting the thermohydraulical *nodes* to the *heat*

structure nodes of the outer surface by using external *heat transfer modules*. Dedicated heat exchangers contain the whole structure in one symbol. This way of discretization is very close to the underlying physical laws, as described below:

- **(Thermohydraulical)** ***nodes*** are used for the conservation laws of mass and energy, so also an average temperature of the fluid in the *node* is calculated.
- ***Heat structure nodes*** represent a discrete part of the tube wall with its heat energy and temperature. This correlation is given by the heat capacity of the corresponding wall material in respect to the temperature $c(T)$.
- ***Heat transfer modules*** read the temperatures of thermohydraulical and *heat structure nodes* and calculate the difference ΔT. ΔT, flow speed, density and other data of the fluid in the thermohydraulical *node* are used to calculate the heat transfer coefficient α. Variations due to fouling, slagging or special geometry can be considered by an efficiency factor, that is multiplied with α.
  The inner or outer radius of the tube, the tube number and their average length are used to calculate the heat transmitting area A and the volume of the calculation node V. As APROS assumes tubes to have cylindrical surfaces, the calculation of A can not be altered.
  The resulting heat flux $\dot{q} = \alpha\Delta T$ is converted into a power density $p = \dot{q}\frac{A}{V}$, which is subtracted from the *heat structure node* and added to the thermohydraulical *node* in respect to their volume. [12]
- ***Heat structure branches*** calculate the heat flow between adjacent *heat structure nodes* from the heat conductivity in respect to the temperature $\lambda(T)$ and geometry data.

2.3 Calculated heat flow

This structure represents well the physical situation. However, it does not give accurate heat flows when it is used in heat exchangers. As in the nodes the calculated temperature is an average value, hotter or colder fluids at the inflow of the heat exchanger that would influence the temperature difference between adjacent nodes are not taken into account. For the design of heat exchangers, the logarithmic temperature difference ΔT_L gives correct values for the heat flow. It is calculated according to formula 1 for a co-current heat exchanger:

$$\Delta T_L = \frac{\Delta T_H - \Delta T_C}{\ln(\Delta T_H / \Delta T_C)} \qquad (1)$$

where:
$\Delta T_H = T_{fluid\_A\_entering} - T_{fluid\_B\_entering}$
$\Delta T_C = T_{fluid\_A\_leaving} - T_{fluid\_B\_leaving}$

As mentioned before, APROS does not recognize heat exchangers as a whole, but only as two adjacent chains

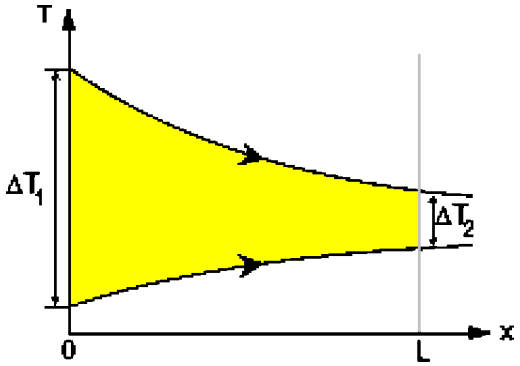

Fig. 4 temperature difference in heat exchanger

of *nodes* that can exchange heat via the adjacent *nodes*. The figs. 4 to 7 demonstrate this effect by showing the real temperature distribution over the flow length of a co-current heat exchanger. Fig. 4 shows the real situation with the driving temperature difference in yellow. Discretizing this heat exchanger in APROS using only one node per fluid results in the situation shown in Fig. 5: Only the temperature difference at the outflow is used to calculate the transferred heat flow which results in a too small value for the heat flow.

Increasing the node number to two nodes per fluid

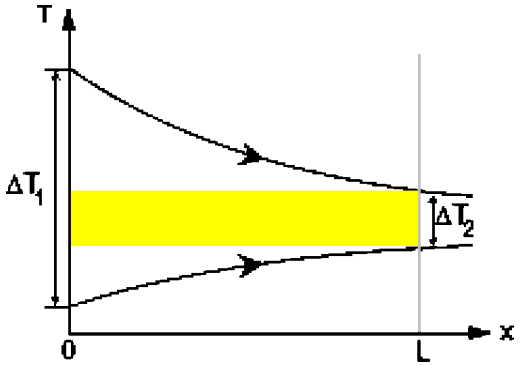

Fig. 5 calculated temperature difference in APROS model discretized in 1 node

results in the situation shown in fig. 6 where the yellow area has come significantly closer to the boundaries. The calculation error in this case is still significant, which is demonstrated by the large white area between the yellow area and the upper and lower boundaries.

As fig. 7 demonstrates, six nodes per fluid have brought the yellow area very close to the boundaries and so reduced the calculation error significantly.
Finding out the optimal number of nodes for the discretization of a heat exchanger is important, as a low value may result in calculation inaccuracies. This is even more grave for counter-current heat exchangers where the heated fluid is heated up to a temperature higher than the outflow temperature of the heating fluid - such as the primary air preheater. It can be modeled satisfactorily with about 20 nodes on each fluid side.

On the other hand, a too large number of nodes will

ISBN 978-3-901608-32-2

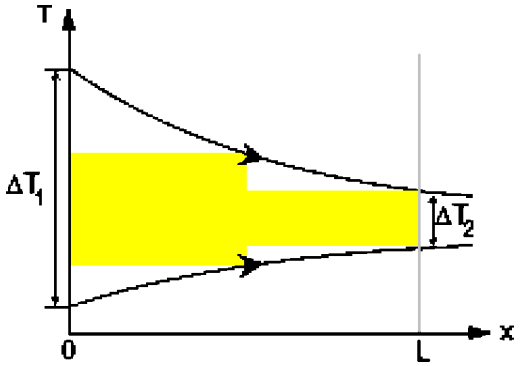

Fig. 6 calculated temperature difference in APROS model discretized in 2 nodes

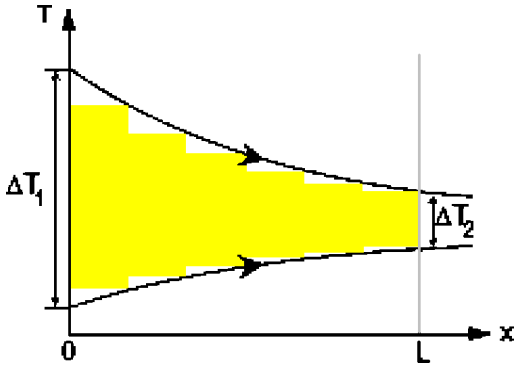

Fig. 7 calculated temperature difference in APROS model discretized in 6 nodes

result in longer calculation times and may also create numerical problems if the individual nodes get smaller than the volume of fluid that is put through them in one time step. As this is solely dependent on the geometry and volumetric flow of this node, a general value can not be given in this place. However, using more than 10 to 20 nodes in a heat exchanger will probably not improve the accuracy of the simulation significantly.

3 Applications in power plants

In power plants, the main components that differ significantly from the tubular geometry APROS is optimized for are finned walls, finned tubes and heat exchangers in preheaters. Methods to model them successfully in APROS are described in the following.

3.1 Finned walls

3.1.1 Application

Finned walls, as shown in fig. 8 are used in almost all fired steam generators to surround the combustion chamber [13]. Due to the large heat flux density from the fire, the finned walls are used as evaporators. Most of the generated heat from the fuel is transferred through finned walls, so modeling their exact behavior in steady-state and during transients is relevant.

As finned walls consist of circular tubes that are connected to each other with bars between them, two different effects for the heat transfer have to be considered: Heat is only transferred on one shell side of the tubes and about one third of the heat is not transferred directly to the tubes, but to the bars and transferred to the tubes by heat conduction.

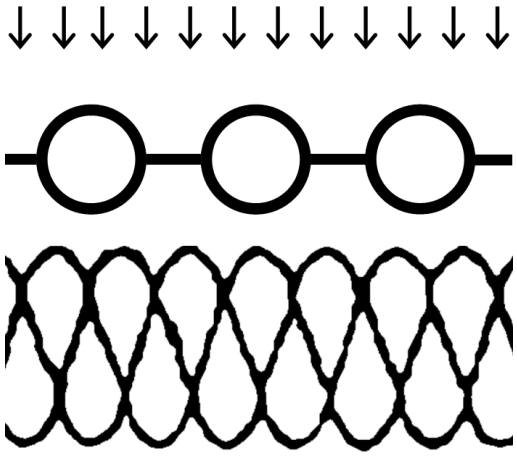

Fig. 8 insulated finned wall, heated by radiation

3.1.2 Discretized model

In the simulation, the first effect can easily be taken into account by dividing the tube wall in two parts: a first and a second side. So one thermohydraulical *node* in a wall heat exchanger can be connected to two different *heat structure nodes* via two *heat transfer modules*. These *heat structure nodes*, representing the two sides of the tube wall can also be connected to further two thermohydraulical *nodes*, representing the fluids at the first and the second shell side. In Fig. 8 the finned wall is heated on one shell side and insulated on the other one - both effects are taken into account in the model.
The effect of the bars between the tubes is more difficult to model because the graphical user interface in the program does not show any possibilities to enter this complex geometry. In most cases, a simplified geometry is sufficient. The proposed adequate modeling is to ignore the bars and increase the heat transfer efficiency on the shell side of the tubes to a proper value. The steel mass of bars can be included by increasing the outer radius of the tubes so that the overall steel mass remains the same.

This simplified model can easily be tuned to give sufficient results for the heat fluxes in steady-state. During transients, the larger heat resistance of the bars is no longer taken into account as the steel masses of the bars are concentrated more closely to the water side. This might make the model react quicker on transients than the real device would.

3.1.3 Validation

For validation, a transient was used that is characterized by very fast cooling of the evaporator, this is realized by changing the position of the turbine valves from the throttled state to full aperture. The thus increased live steam mass flow is followed by an adequate reduction

ISBN 978-3-901608-32-2

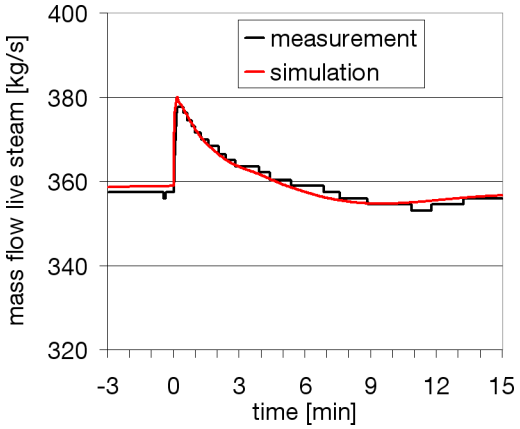

Fig. 9 live steam mass flow after releasing throttling of the steam generator

of the pressure in the steam generator. In the evaporator, the temperature of the water equals the saturation temperature which also falls according to the falling pressure. Due to the efficient heat transfer on the water side of the evaporator, the wall temperature falls as well with almost the same speed. The stored heat energy in the wall is partially transferred to the water side, increasing the steam production. Prior simulations have shown significant correlation of the live steam mass flow during this transient and the steel masses of the evaporator in the steam generator. In a coal-fired power plant measurement data on this transient was taken and used to validate this simplified model of the finned wall.
Fig. 9 shows the live steam mass flows measured and taken from the simulation model of the same power plant. The plant was operating at about 90 % of its rated power, which is 450 MW, in modified sliding-pressure operation. At $t = 0$, the throtteled turbine valves were fully opened with maximum speed and kept at full aperture for the rest of the transient. Unit control and fuel control were switched to manual operation. The rise of the steam mass flows, the decline according to the capacity of the boiler and the steady-state is clearly visible. The good correlation between measurement and simulation shows the adequacy of this simplified finned wall model also for the simulation of fast transients. More detailed discretization of finned walls in evaporators will not be necessary in most cases. For special cases, a similar discretization scheme as described in Chapter 3.2 is proposed.

3.2 Finned tubes

3.2.1 Application

Combined-cycle power plants have become very common due to their high efficiency. Heat recovery steam generators (HRSG) combine the gas and the steam cycle and so are the main element in combined-cycle power plants. They produce steam with temperatures of about 550 °C from the exhaust heat of the gas turbine, which is at about 600 °C. The smaller this temperature gap is, the higher is the efficiency of the overall cycle. As there is almost no heat transfer by radiation due to the relatively low temperatures, the conductive heat transfer must be optimized. The heat transfer on the water and steam side of the HRSG is much more efficient than on the flue gas side, so the latter is to be improved. This can be done by increasing the heat transmitting area on the flue gas side. Usually, finned tubes, as shown in Fig. 10 are used for that purpose [14] so the heat transfer takes place mainly on the surface of the fins and only to a minor degree at the outer surface of the inner tube. Due to their complex geometry, they do not have a detailed component model in the simulation system either.

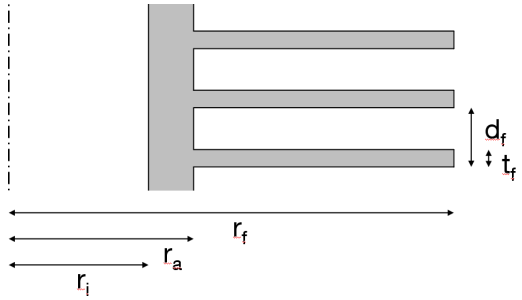

Fig. 10 finned tube

3.2.2 Demands on the model

For an accurate calculation of the dynamics of finned tubes, a simple pipe model is not sufficient [15]. The component model shall consist of a segmented model of a finned tube, that takes into account the distributed heat transfer from the gas on the fins in radial direction, the heat capacity of the fin material and the heat conduction over the fins in radial direction. The distributed radial temperature of the fin also influences the radial heat flux density, which shall also be taken into account.
On the flue gas side of an HRSG the flow is usually laminar. Differences in the local flue gas temperatures at the same cross-section may occur and are to be considered in the model.

3.2.3 Constraints

Due to the complex geometry of finned tubes, the following constraints have to be made:

- Axial discretization only respects differences in the local water and flue gas temperatures. Axial heat conduction in fins or tubes is not recognized.
- Discretization of tubes and fins is implemented only radially, not tangentially or axially.
- The heat transfer coefficient α is locally constant for all heat transmitting surfaces on the flue gas side on the same section of one pipe.
- There is no heat transferred by radiation. Due to the low temperatures in a HRSG, this is almost correct.

ISBN 978-3-901608-32-2

- The flue gas pressure drop due to the finned pipes is not relevant.

3.2.4 Modifications

The modularity in the underlying calculation level of the program offers the possibility to create a model that fulfills the requirements mentioned above: As described in chapter 1.4, tube walls may consist of up to three different materials. Any of these layers can be further discretized into several finite volumes. The heat transfer from the inner or outer tube side to the inner or outer fluid *node* is done by *heat transfer modules*. Modifying this discretization to represent a finned tube can be done as explained in the following steps:

Material definition

The finned tube model is a modified *pipe with heat structure*. As this pipe model is massive, it is configured with two layers, where the inner layer corresponds to the tube of the heat pipe and the outer layer corresponds to the fins. Usually in finned tubes different kinds of alloy are used for the tube and the fins, so two different materials must be defined in the material database to take into account the different heat capacities and conductivities.
To recognize the distance between the fins in this model, heat capacity $c(T)$ and heat conductivity $\lambda(T)$ of the fin material must be corrected. This is done by decreasing the values for heat capacity and heat conductivity of the fin material in respect to the fin density as shown in formulas 2 and 3.

$$\lambda_{database}(T) = \lambda_{material}(T)\frac{t_f}{d_f} \tag{2}$$

$$c_{database}(T) = c_{material}(T)\frac{t_f}{d_f} \tag{3}$$

As this material is only valid for a certain density of the fins, changes in fin material or fin density require additional or modified material definitions.
Abstained from the small axial heat flux at the fin ground, this model is sufficient for realistic calculations of radial heat flux and storage in finned pipes.

Radial discretization

The desired segmented model of a finned pipe that discretizes the pipe radially can easily be configured by increasing the number of *heat structure nodes* in the tube and fin layer. A number of four nodes in the tube layer and five nodes in the fin layer is regarded as sufficient. As the complexity of the model would significantly grow, the discretization is not increased any further. Auxiliary calculations on the radial temperature profiles in the fins with five and fifteen nodes show very small differences, as fig. 11 shows.
Fig. 12 demonstrates the radial heat flux in the fins, in respect to the radius. Due to the different discretization grid, the results are in parallel but show the same characteristics. The points at the very left, that cumulate all the heat flows from the fin practically show the same heat flux. A remaining error of less than 2 % is almost

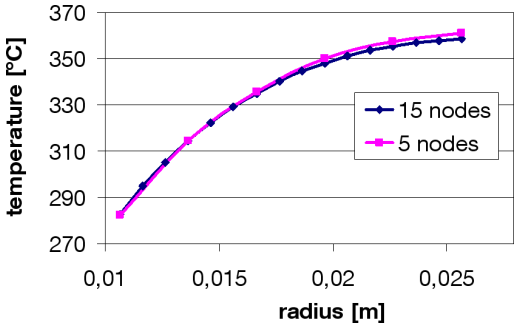

Fig. 11 calculated temperature profile of fins

negligible, but can easily be regarded as a correction factor to be respected later. (Formula 4)

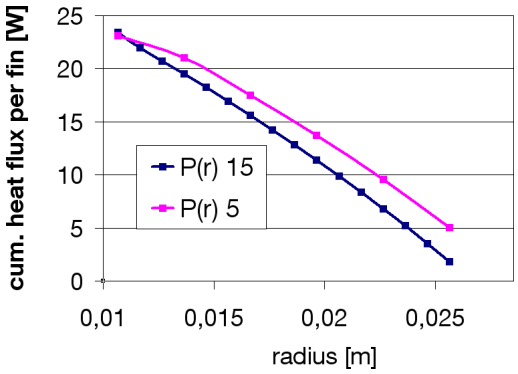

Fig. 12 calculated heat flux profile of fins

$$C_{disc\_rad} = 1.02 \tag{4}$$

Calculation errors due to low discretization in flow direction

The temperature of the flue gas may change significantly when passing only one tube row in a HRSG. As one tube row equals one node, the heat flow is calculated too low, as described in 2.3. The error differs from tube row to tube row in respect to the relative temperature change. For simplifications, an average error for the whole HRSG of 10% was calculated, which is respected in another correction factor (5).

$$C_{disc\_flow} = 1.10 \tag{5}$$

The dependency of C_{disc_flow} from the local heat fluxes can additionally be taken into account: At the beginning of the evaporator section, where very large temperature differences and heat fluxes occur between flue gas and water/steam, the factor shall be significantly larger that close to the pinch point section.

Additional heat transfer modules

As mentioned before, heat transfer in APROS only takes place at the inner and outer surface of a tube. Manual definition of *heat transfer modules* to any *heat nodes* is also possible. To take into account the radial

heat transfer on the fin surface, additional *heat transfer modules* are manually defined that connect not only the *heat nodes* on the outer shell side to a *node* representing the shell side, but all of the *heat nodes* in the fin area. Fig. 13 demonstrates the arrangement. The outer *heat node* of the tube layer can also be connected to that flue gas *node*.

These *heat transfer modules* calculate the heat transmitting area as the surface of a cylinder. For one fin, formula 6 demonstrates the value APROS uses. The real heat transmitting area of one fin mainly consist of annuli and only few cylindrical parts (formula 7). To get the transferred heat right, the area ratio is respected in another correction factor C_A, according to formula 8.

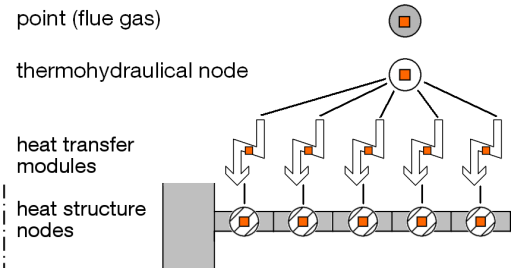

Fig. 13 heat transfer modules in finned tube model

$$A_{APROS} = 2\pi r_{heatnode} d_f \quad (6)$$

$$A_{real} = 2\pi(r_f^2 - r_a^2) + 2r_f\pi t_f + 2r_a\pi(d_f - t_f) \quad (7)$$

$$C_A = \frac{A_{real}}{A_{APROS}} \quad (8)$$

C_A has to be calculated and applied for all of the radial *heat transfer modules* individually.

Local heat transfer coefficient

The gas flow on the surface of a finned tube differs significantly from the gas flow around a straight tube. For this reason, the internal formulas for the heat transfer coefficient on the flue gas side can not be used directly for finned tubes.

It is assumed that the used Dittus-Boelter equation (formula 9) [10] [12] calculates the changes of the heat transfer coefficient in respect to the flue gas velocity and other flue gas related parameters correctly.

$$\alpha_{DB} = 0.023(\frac{\lambda}{D_{eq}})Pr^{0.4}Re^{0.8} \quad (9)$$

So the relative error to the real heat transfer coefficient can be compensated by multiplication with one factor C_α that is assumed to be constant. For this calibration a test facility on modern HRSGs of the Institute of Energy Systems at TUM (Technische Universität München) provides data using an established program for the design of HRSGs. These calculated values for the heat transfer coefficient are compared to the values taken from APROS according to formula 10.

$$C_\alpha = \frac{\alpha_{design\_program}}{\alpha_{DB}} \quad (10)$$

When exporting the heat transfer coefficient from APROS, the constraints (working point, flue gas speed, temperature, ...) in the APROS model must be the same as in the layout program. So a realistic heat transfer coefficient at the flue gas side of heat pipes can be achieved.

Heat transfer modules in APROS can be tuned by multiplying the calculated heat flow with an extra value. The four correction factors give one by multiplication according to formula 11 which is applied in APROS.

$$efficiency_{APROS} = C_\alpha C_A C_{disc\_rad} C_{disc\_flow} \quad (11)$$

Review

Due to the very complex surface of a finned tube, the heat transfer coefficient is not constant on the surface, there are significant changes between the front and the back side of the tube. Reliable values for the local heat transfer coefficient can only be obtained by CFD-calculations that discretize the fins and the flue gas volume in all three dimensions.

As this is not possible in APROS and requires too much calculation time for calculations of plant dynamics, this simpler approach is performed. Slightly reduced accuracies must be accepted.

The significantly altered geometry on the flue gas side also interferes the calculation accuracy of the pressure loss of the flue gas in the HRSG. Due to the very low pressure loss in most HRSGs, this effect may be neglected in most cases.

Axial discretization

In one pass of a economizer or superheater tube, water or steam can receive a significant rise of temperature. This may lead to different temperature differences between flue gas and water/steam in the same cross-section of the flue gas channel. To reduce resulting errors of the local heat flow, the flue gas channel is split to ten separate pipes in parallel order. As the gas flow in the flue gas channel is usually laminar, there is only few mixing of hotter and colder parts of the flue gas in the flue gas channel and this approach is valid. It also offers the possibility to respect gradients in flue gas speed or flue gas temperature that are caused by the gas turbine, auxiliary gas burners or the diffuser of the gas turbine.

The finned tubes are discretized axially to ten sections. On the water/steam side, this improves the calculation accuracy of the local heat transfer coefficient, as pipe sections with boiling water are better separated from sections with supercooled water or superheated steam. The *nodes* in the flue gas paths and the *heat structure nodes* in the finned type model are manually connected by *heat transfer modules*. Fig. 14 demonstrates a simplified structure. The boxed background marks the components belonging to the finned tube model: The pipe is axially discretized in thermohydraulical *nodes* and radially discretized in *heat structure nodes*. Ten symbols of flue gas pipes on the right side demonstrate the discretization there. The *heat transfer modules* are shown as red lines that connect the nodes in the flue gas pipes with the corresponding *heat structure nodes* in the finned pipe model.

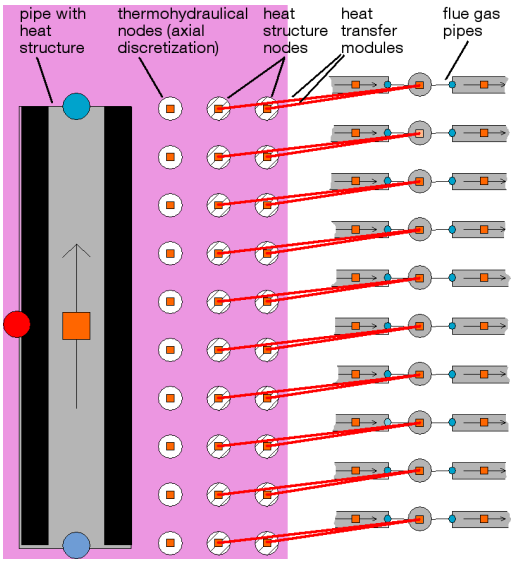

Fig. 14 heat transfer from finned tube model to flue gas pipes

3.2.5 Validation

Using this approach, a segmented model of a finned pipe can be achieved. As the performed modifications only apply on heat transfer calculations which do not affect plant dynamics, validation of the stationary state is regarded to be sufficient. This can be done by calculating the temperature increase of the fins in radial direction in steady state manually. In fig. 15 the simulated and manually calculated temperature profile is displayed. A temperature difference of about 60 K between foot and top of the fins is visible in both calculations. The temperatures at the foot point are adjusted to the same value. Differences between the calculations in radial direction are almost negligible. On the fin top, the calculation error is maximal but does not exceed 1 K. So the proposed method does not only prove the ability of APROS to implement a segmented model of a finned type but can also give accurate results, if proper values for the heat transfer coefficient are available.

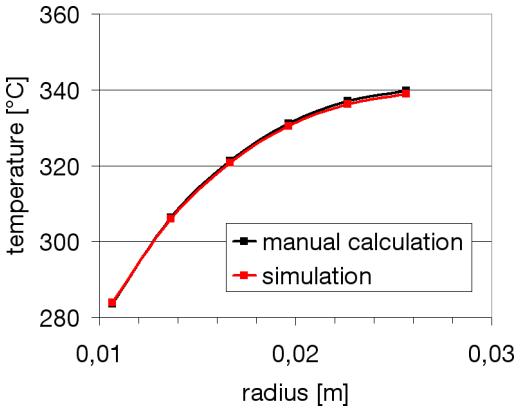

Fig. 15 Calculated and simulated radial temperature profile

3.3 Heat exchangers

3.3.1 Application

Two well known kinds of heat exchangers in power plants have been described in the previous chapters, as their discretization is more complicated due to their geometry. There are also heat exchangers in power plants that consist of straight tubes or plates where APROS has adequate formulas for. In this cases, it is assumed that the flow on the shell side moves axially and the local values for Reynolds number, temperature and flow speed are calculated correctly in the one-dimensional discretization in APROS. In the heat exchangers described below, this is not exactly the case which makes some heat exchanger bundles in power plants are difficult to simulate exactly. In the following, the most significant heat exchangers are discussed.

Superheaters and reheaters

Superheaters and reheaters are located in the flue gas path. As modern firing systems use tangential burners the gas flow in one cross-section shows a significant rotating. Flue gas speeds and heat transfer coefficients in the outer areas of superheaters and reheaters are significantly higher than in the centre which creates very different steam temperatures at the end of a superheater. [16] Due to the one-dimensional approach in APROS, these detailed effects can not be simulated. Only the overall heat flow in a superheater or reheater can be corrected and adjusted.
For that purpose the efficiency in the heat transfer module must be increased. Values of $1.2 < efficiency_{superheater} < 1.8$ are considered to give realistic results. To minimize discretization errors as described in 2.3 counter- or co-current heat exchangers with about five to ten *nodes* per fluid section give better results than cross-flow heat exchangers.
In fired steam generators, fouling and slagging are additional phenomena that decrease local and overall heat transfers. This can also be respected by an additional decrease of the efficiency factor.

Pre-heaters

Pre-heaters are heat exchangers in power plants that use tapped steam from the steam turbine to increase the temperature of the feed water before entering into the steam generator. They are constructed as tube bundle heat exchangers where the feed water flows inside the tubes and the steam enters at the shell side and is desuperheated, condensated and sub cooled on the outer surface of the tubes. The by far largest amount of heat is transmitted in the condensation process. As in Nusselt's heat transfer calculation for condensation on the surface of horizontal pipes steam velocities are not relevant (formula 12) [17],

$$Nu = 0.725 \sqrt[4]{\frac{gr\rho^2 l^3}{\eta\lambda(T' - T_0)}} \qquad (12)$$

the calculation of the condensation phase is very stable and accurate. In the sub cooler section this is not the case as local flow velocities and Reynolds numbers

influence the heat transfer significantly. As APROS supposes straight axial flow in the sub cooler on the shell side, the transferred heat is calculated too low as the significant turbulences in the hotwell are not recognized. Sub coolers do not transfer significant amounts of heat and do not either show any relevant behavior during load changes, so their effect on overall plant performance is rather small. So in most case studies it is no problem to adjust the heat transfer efficiency of the sub cooler manually to an approximative value. Values of $efficiency_{subcooler} \approx 4$ are considered to give suitable results.
The specific volumes of low-pressure steam in the first stages of the pre-heater section and cool water differ for about four orders of magnitude. This may have significant influence on the stability of the simulation. To increase the convergence of the numerics, two methods are proposed:

- Structures with sub cooled water under low pressure (sub coolers of the first pre-heater stages) should be placed far below the corresponding pre-heaters. The elevation difference increases the supercooling of the water and vaporisation is less likely to appear.

- As sub coolers usually transfer only small amounts of heat, compared to the condenser sections of the pre-heaters, omitting them in the simulation model may also be a suitable solution.

4 Conclusion

As an accurate modeling of all the components in a power plant is not possible due to the system complexity, the user has to model it as a reduced order system with adequate simplified components. Inaccuracies in minor component models hardly affect the behavior of the whole plant, so they can often be ignored if the specific component is not of particular importance for the studies. As presented in the previous chapters, this may be the case in sub coolers.
For more relevant plant components, APROS offers the possibility to alter the discretization and apply further correction factors manually so that better conformity can be achieved. This will also significantly improve the results of the simulation. This was described in detail for finned pipes in HRSGs.
Due to the complexity and variety of conducted power plants and the multiple configuration options APROS offers, the user is faced two main demands: He must have detailed knowledge on the discretization scheme of the program and he has to know the plant internals thoroughly. Only with detailed design data he is able to estimate the influence of the components, to distinguish minor from major components and to implement the appropriate discretization for all of the components. In this case, modern system shells for numerical simulation of plant dynamics can provide valuable information. Manufacturers as well as utilities can use the software with only small efforts on time and costs.

5 References

[1] http://www.grs.de.

[2] http://www.grs.de/en/simulationcodes/athlet.html?pe_id=381.

[3] GRS. *Overview of ATHLET Mod 2.1 Cycle A*. http://www.grs.de/module/layout_upload/overview.pdf, April 2001. download: 23.4.2007.

[4] http://www.vtt.fi.

[5] http://www.vtt.fi/apros.

[6] Technical Research Centre of Finland. *APROS*. http://www.vtt.fi/apros/pdf/general/general - apros overview.pdf. download: 23.4.2007.

[7] http://www.mathworks.com.

[8] E. Klotz. Comparison of two software tools for the dynamic simulation of power plants. Master's thesis, Lehrstuhl für Energiesysteme, TU München, Garching, 2006.

[9] P. Colonna. Dynamic modeling and simulation of energy conversion systems. Lecture notes, Delft, 2006.

[10] M. Hänninen. A solution of one-dimensional flow and heat transfer processes with difference method. Master's thesis, Department of Energy Technology, Lappeenranta University of Technology, Helsinki, 1988.

[11] VTT. APROS Documentation.

[12] J. Ylijoki M. Hänninen. The constitutive equations of the apros six-equation model. Technical report, VTT, April 2006.

[13] R. Dolezal. *Dampferzeugung: Verbrennung, Feuerung, Dampferzeuger*. Springer, Berlin; Heidelberg; New York; Tokyo, 1985.

[14] Ch. Pletl. *Experimente und numerische Simulation zum dynamischen Verhalten eines Abhitzedampferzeugers mit Zwangdurchlauf*. VDI, Düsseldorf, 2005.

[15] Z. Hu. *Zeitverhalten einer Kraftwerksanlage am Beispiel eines Gasturbinen-Dampfturbinen-Kombiprozesses mit Zwangsdurchlauf-Rippenrohr-Abhitzekessel*. VDI, Düsseldorf, 1993.

[16] Ch. Sauer et al. Detaillierte gekoppelte Simulation von Großkraftwerken. *VGB-Konferenz*, 2000.

[17] Beitz W. and Küttner K.-H., editors. *Taschenbuch für den Maschinenbau / Dubbel*. Berlin, Heidelberg, New York, 14th edition, 1981.

