HEAT TRANSFER COEFFICIENTS IN A TURBOCHARGER WITH AND WITHOUT BOILING

Thomas-Sebastian Gier  
RWTH Aachen  
Institut für Kraftwerkstechnik,  
Dampf- und Gasturbinen  
gier@ikdg.rwth-aachen.de  
52056 Aachen, Germany

Thomas Biesinger  
BorgWarner Turbo Systems GmbH  
Global Engineering Core Sciences  
tbiesinger@borgwarner.com  
67292 Kirchheimbolanden, Germany

Henning Eickenbusch  
ANSYS Germany GmbH  
henning.eickenbusch@ansys.com  
83624 Otterfing, Germany

ABSTRACT
A robust approach to the computation of heat transfer with and without boiling is presented. The thermal boundary layer thickness is identified by both temperature gradient and shear stress transport methods that coincide on almost identical isosurfaces. Heat Transfer Coefficients (HTCs) and their corresponding fluid temperatures without boiling have to be determined by a diabatic Computational Fluid Dynamics (CFD) computation when a Conjugate Heat Transfer (CHT) model is not available. They are obtained by dividing the wall heat flux by the temperature gradient from the wall to the isosurface. A single CFD/FEA (Finite Element Analysis) iteration is required to provide thermal boundary conditions for subsequent life time part assessments. Another objective of the present work is to demonstrate the capability of an engineering nucleate wall function based model after Chen to predict boiling at low computational cost. The model is calibrated and validated against a range of test cases of different fidelity: a heated channel, a turbocharger with a cooled aluminium turbine housing and a bearing housing with a water core. Here, the HTCs are obtained from a CHT computation with the boiling model by addition of convective and boiling heat fluxes.

INTRODUCTION
Heat transfer is important for turbomachinery applications such as turbochargers. Its accurate prediction enables the design of highly efficient engines. The Environmental Protection Agency as well as EU regularity bodies have been implementing increasingly restrictive emission standards to limit pollutants emitted by vehicles. Turbochargers play an important role in meeting these standards.

One objective of the present work is to obtain boundary conditions for a Finite Element Analysis (FEA) to determine material stress. For this purpose, heat transfer coefficients, fluid and solid temperatures need to be extracted from a Computational Fluid Dynamics (CFD) simulation. The software used in this paper is ANSYS CFX.

A non-dimensional HTC is customarily defined as heat flux divided by its driving temperature difference:

\[ HTC = \frac{q}{T_{\text{solid}} - T_{\text{fluid}}} \]  

(1)

The fluid temperature near a wall can be difficult to extract from simulations i.e. using a local HTC defined via the fluid temperature at the edge of the boundary layer leads to a more robust simulation. It should be noted that the present investigation refers to static values as obtained from CFD. Textbooks (Schlichting, 1968; Baehr and Stephan, 1994) suggest defining the displacement thickness of a fluid or thermal boundary layer at about 99% of their respective freestream value. A related approach to identify the thermal thickness by means of looking at the temperature gradient at a specific non-dimensional boundary layer thickness is presented. The resulting HTCs coming from a diabatic CFD run are then only a function of the geometry (VDI Wärmeatlas, 2013). They are valid for both laminar and turbulent flow.

The current standard in industry is to obtain HTCs from an iteration of a diabatic CFD and FEA solution. An adiabatic run with a constant fluid temperature is used to start
the process prompting for further iterations. Schmidt and Starke suggest an internally coupled CFD/FEA approach to avoid outer iteration loops (Schmidt and Starke, 2015). The method presented below requires a single iteration, if fluid temperatures or reverse heat fluxes are to be computed in close agreement with conjugate heat transfer (CHT) results. This approach is limited to single-phase fluids only, like the exhaust gas in a generic turbocharger. For cases where the turbocharger is water-cooled the effects of boiling have to be taken into account, namely an increased heat flux. Importantly the HTC varies with the wall temperature and necessitates a CHT simulation.

A combination of two modelling approaches have to be added to the simulation to predict boiling:

1. A multiphase model to account for the simultaneous presence of both liquid and vapor
2. A wall boiling model to account for the increased heat flux due to boiling

The simulation switches to wall functions when the boiling model is used. The resulting decrease in heat flux is compensated by a mesh correction factor. A theoretically possible methodology for LRN meshes is suggested below. Robustness i.e. convergence of the simulation including boiling requires additional attention.

**THERMAL BOUNDARY LAYER**

The Diabatic Temperature Gradient (DTG) and the Diabatic Y-plus Field (DYF) methods are an improvement over the industry standard approach that mostly relies on adiabatic fluid temperatures. The key element of both of the present methods is the fluid temperature to be known at the edge of the thermal boundary layer. Even if the solid temperature is kept constant in a first diabatic run, valid HTCs can be obtained as the heat flux and temperature difference partially cancel (Eqn 1). A schematic of the overall process to obtain HTCs and fluid temperatures for exhaust gas and cooling water is depicted in Figure 1. The thermal boundary based methods are also shown to lead to more physical HTCs in subsequent iterations.

**Model Setup**

For the purpose of developing the methods presented in this paper the turbine of a turbocharger with an aluminium turbine housing is used. This allows for two different simulations to be done on the same model. The development of the DTG and DYF methods takes place using only the hot gas path of the turbine, which contains the volute, turbine, diffuser and wastegate. The inclusion of the boiling model is tested on a full CHT model of the entire turbine, including the hot gas path, water core and solid bodies. Figure 2 depicts the turbine that is used.

The hot gas path includes the full turbine wheel and is modeled using 5.6 million nodes. A frozen rotor interface is employed to capture the influence of the volute on the wheel and flow separation going into the diffuser. Turbulence is modeled using the SST turbulence model under the assumption that the flow is fully turbulent. Dimensionless wall distance y-plus is kept below one in order to use a low Re for boundary layer modeling. The exhaust gas is modeled as an ideal gas.

For the CHT simulations the water core and housing with three hundred thousand, respectively two hundred fifty thousand nodes, are added. The water core uses a wall function approach for the boundary layer due to requirements stated in the upcoming section “Computational Mesh”. Accordingly the y-plus value is kept above 20 and below 100. To account for the possible phase change in the water core two materials are defined. The liquid phase uses a mixture of 50% water and 50% ethylenglycol as commonly used in automotive radiators while the vapor phase consists of 100% water. This is due to the lower boiling point of water, which results in only a negligible fraction of ethylenglycol vaporizing under the conditions in the water core. Properties for these materials were obtained from Robinson (Robinson, 2001). The solid-state material properties are assumed to be constant.

**Diabatic Temperature Gradient Method**

In the Diabatic Temperature Gradient (DTG) variation of the method a constant value of the temperature gradient is used to identify the boundary layer edge. As can be seen in Figure 3 the temperature gradient is highest near the wall and decreases with further distance. Once it reaches the defined value the corresponding coordinate is identified as the edge of the thermal boundary layer.
This method is easy to implement, since the temperature gradient can be calculated in post-processing from the temperature values already present in the results. The disadvantage is a requirement for very small grid cells near the wall, because otherwise the resolution of the temperature gradient is too low to properly identify the edge of the thermal boundary layer. Thus, this method is only suitable for meshes with fine near-wall resolution like is required for a Low Reynolds Number (LRN) approach to modeling near wall flow.

An alternative to the DTG method is the Diabatic Y-plus Field (DYF) method. This approach is based on relating the thermal boundary layer thickness to the non-dimensional fluid boundary layer thickness $y^+$. The Reynolds analogy states identical values for Prandtl numbers of unity. However, the analogy does not hold under non-equilibrium conditions i.e. short and non-flat geometries dominated by convection. In the present study, the thermal thickness is found to constitute a fraction of the fluid boundary layer thickness by consideration of its temperature gradient. Karamavruc quotes a freestream value for $y^+$ of 250 characteristic for fluid boundary layers in turbochargers (Karamavruc, 2016). A value of $y^+$ 80 captures about 90% of the range of the temperature gradient encountered in the examined turbine housing, Figure 4. This value is used in subsequent investigations as a proxy to define the thickness of the thermal boundary layer.

The DYF method relies on the availability of $y^+$ in the entire fluid domain. $y^+$ values are customarily available only next to the wall. However, ANSYS CFX provides a means of implementation as an approximate field variable.

The implementation is tackled by calculating equation (2) for every vertex. The only limitation here is the fact that wall shear stress is only available at grid vertices on the wall. To work around this, a Poisson Equation is added to the simulation:

$$\nabla(\rho D_\phi \nabla \phi) + S_\phi = 0$$

Diabatic Y-plus Field Method

Figure 2 Typical Turbocharger with Cooled Turbine Housing

Figure 3 Near Wall Fluid Temperature Gradient at Various Turbine Housing Locations

Figure 4 Near Wall Fluid Temperature Gradient at Various Turbine Housing Locations

Figure 5 Overlay of Isosurfaces Representing the Thermal Boundary Layer for DTG (green) and DYF (blue) on Turbine Housing and Volute
This transports the value of the wall shear stress perpendicular to the wall in a manner similar to a diffusion process. To accomplish this a Dirichlet boundary condition is set on all walls with the value of the wall shear stress. All other boundary conditions are Neumann boundary conditions with the value zero.

The advantage of this approach is the possible application on coarse near-wall grids, as used in simulations which employ wall functions to model near-wall flow. In addition the surface formed by the locations of all identified locations in the DYF method has been found to produce more robust results, especially in areas with complex flow patterns.

The disadvantage of this method is the addition of another partial differential equation to the simulation, which cannot simply be added in post-processing without requiring additional iterations in the solver. Furthermore it increases the simulation time by about 15%.

**Diabatic Results**

The isosurfaces leading to the fluid temperature as obtained from DTG and DYF are found to coincide mostly. The DTG isosurface for the volute resides just underneath the DYF in Figure 5. HTCs are then computed by interpolation of the fluid temperatures from the isosurface onto the geometry. Structural mechanics life time assessments require both fluid temperatures and HTCs as boundary conditions.

A diabatic computation assuming constant wall temperatures is usually conducted first. To achieve higher accuracy and account for correct reverse heat fluxes a single CFD/FEA iteration is required. As seen qualitatively (constant range of variables) in Figure 6 there is a negligible difference between the first iteration of the DTG method and the CHT model. Thus, the DTG/DYF methods can provide significant time-savings with turn-around times of one working day.

For simulations with boiling the only major difference is in the setup as a CHT simulation with a wall boiling model. As a result the total heat flux, comprised of a convective and a nucleate boiling component, has to be used for calculating HTCs. Otherwise the calculation of fluid temperatures and HTCs follows the same approach as the DTG/DYF methods.

**SIMPLIFIED BOILING MODEL**

Cooling fluids in a liquid state are subject to boiling, which is the rapid vaporization of fluid once it is heated past its boiling point. The effect of wall boiling on wall heat transfer is characterized by several distinct boiling regimes depending on temperature. These regions are most easily described by utilizing the Nukiyama diagram, which shows the wall heat flux as function of the wall temperature. As can be seen in Figure 7, without boiling the wall heat flux is a linear function of the temperature, corresponding to a constant heat transfer coefficient (Point A to B'). Once nucleate boiling begins the wall heat flux rises exponentially, which represents an increase in the heat transfer coefficient. This effect can be utilized for engineering purposes, since it enables a better heat transfer using less resources (Point B' to C). The limit of this approach is the critical heat flux, which marks the local maximum of the wall heat flux. At this point (Point D) the wall surface instantly becomes covered by vapor, which insulates the wall and causes a massive rise in temperature, which usually results in the destruction of the component (Point D'). An accurate wall boiling model is therefore required to take advantage of the enhanced heat transfer properties offered by boiling effects without causing damage.

**Figure 6 Diabatic CFD w/ Constant Solid Temperature (left), First Iteration w/ FEA Solid Temperature (mid), CHT Model (right)**

**Figure 7 Nukiyama Diagram**

In the study presented here the so-called “Simplified Boiling Model” (Das and Punekar, 2013) based on the work of Chen (Chen, 1966) was used to accurately predict boiling.
The model was translated from ANSYS Fluent to ANSYS CFX and subsequently calibrated.

The reason for choosing the Simplified Boiling Model was its simplicity, and thus numerical efficiency, in addition to its capability to represent the effect of fluid flow on the boiling heat transfer.

Possible alternatives include the Rohsenow model, which has a similar basis to the Chen model, but does not capture flow effects (Rohsenow, 1953), and the RPI model (Kurul and Podowski, 1991). In contrast to the other two options the RPI model employs a more complex approach by using various submodels to represent the phenomena, which occur during wall boiling. Besides being more computationally expensive this approach is also limited by the fact that the RPI model was developed for nuclear reactors and is thus only suitable for pressures far in excess of those employed for the cooling of a turbocharger.

A simulation including boiling necessarily has to account for the simultaneous presence of a liquid and a vapor phase. For this purpose, the SBM employs a homogenous mixture model as its multiphase model. This model tracks the volume fraction of both phases via an additional variable. Relative motion between the phases is considered negligible, because the vapor bubbles only form in close proximity to the surface and condense quickly.

The transfer from liquid to vapor phase is calculated according to the Lee model:

\[ \dot{m}_{l\rightarrow v} = C_{evap} \varphi_p \left( \frac{T_r - T_{sat}}{T_{sat}} \right) \]  

\[ \dot{m}_{v\rightarrow l} = C_{cond} \varphi_p \rho_v \left( \frac{T_r - T_{sat}}{T_{sat}} \right) \]  

As basis for the wall boiling model the correlation developed by Chen (Chen, 1966) was chosen. It treats the effects of fluid flow, from bulk to local variables.

The convective heat flux is multiplied with a correction factor to account for the fact that the vapor content in the mixture as a result of boiling, inhibits the convective heat transfer compared to the pure liquid.

As further part of the validation process the wall heat fluxes calculated with a coarse grid and wall functions were compared to a fine grid with LRN approach, as well as experimental values.

COMPUTATIONAL MESH

During the validation process of the SBM it was discovered that the model requires a minimum wall distance of the first grid node to work reliably. This is caused by the part of the multiphase model, which calculates the mass transfer from the liquid to the vapor phase. The equation is valid only for bulk temperatures. In the case of small wall distances the temperature in the grid cell is not the bulk temperature of the entire vapor bubble, but instead the local temperature of a small portion of it, see Figure 8. Since the local temperature is much higher next to the wall an unrealistically high mass transfer from liquid to vapor phase is calculated, which crashes the simulation.

To resolve this issue, it is proposed that the mass transfer between phases near the wall should not be calculated for every grid cell individually. Instead the mass transfer should be calculated for a “virtual” cell with a height equal to the diameter of a vapor bubble. The resulting mass transfer can then be split among the constituting cells according to their volume. An implementation of this approach is envisaged but beyond the scope of this paper. Instead, a wall distance in the order of the expected bubble diameter was chosen.

WALL HEAT TRANSFER

As further part of the validation process the wall heat fluxes calculated with a coarse grid and wall functions were compared to a fine grid with LRN approach, as well as experimental values.

The test-case used for this process is the test rig employed by Robinson for his experiments concerning boiling heat transfer (Robinson, 2001). This test-case was already used by Das and Punekar to validate their model (Das and Punekar, 2013) and thus makes a good point of comparison.

The layout of the test-case is shown in Figure 9. It consists of a straight channel with a height of 10mm, width of 16mm, and a length of 241mm. 76mm downstream from the inlet a heating surface is located with a width of 10mm and a length of 50mm. The fluid used was a 50:50 mixture of water and ethylenglycol, and inlet velocity, pressure and the temperature of the heating surface was varied.

Fluid Temp.

Figure 8 Bubble in Wall Function Grid (left), LRN Grid (right)
Several out-of-the-box simulations without the boiling model were conducted to study the impact of the near-wall treatment with wall functions against the LRN approach. The simulations were run at operating points for which the data from Robinson indicated no boiling. One such data point is shown in Figure 10 with an inlet velocity of 0.5 m/s, a pressure of 3 bar and surface temperature of 124.8°C. The results indicate a significant discrepancy between simulation and experiment. E.g., the wall heat flux calculated with the LRN approach is only about half the value measured by Robinson. Results by Das and Punekar (2013) that are not shown here reach generally good agreement although wall functions are used. Their results included an additional turbulent eddy viscosity added to the dynamic viscosity $\eta_m$ in the correction factor $F_{corr}$, see Eqn. 7. It is found that this modification leads to divergence in most geometries more complex than a straight channel.

The present work presents measures to obtain robust predictions with the SBM for a wide range of applications. Firstly, the well-known influence of the mesh, which is due to the lack of the boundary layer resolution, needs to be taken into account as detailed below. Secondly, the correction factor $F_{corr}$ is reset to its original value in order to stabilize the convergence and preserve the physics. A combination of these two factors leads to a fair agreement i.e. underprediction of the measurement, Figure 11.

Simulations without boiling for which the correction factor $F_{corr}$ is not active still underpredict the experiment by about half. However, the heat flux is an order of magnitude lower than found in realistic applications. It was decided not to temper with the calibration of a state-of-the-art solver. The following section with applications confirms the capability to indicate the correct trend and to obtain convergence.

The mesh adaptation factor for coarse or wall function and LRN wall treatments is added as a multiplier in the convective portion.

$$q_{total} = q_{conv} \cdot F_{corr} \cdot F_{mesh} + q_{nh}$$  \hspace{1cm} (10)

The straight channel test case predictions, Figure 10, were used to calculate the $F_{mesh}$:

$$F_{mesh} = \frac{\alpha_{LRN}}{\alpha_{WallFct}}$$  \hspace{1cm} (11)

The resulting mesh adaptation factors $F_{mesh}$ shown in Table 1 were applied in the remainder.

<table>
<thead>
<tr>
<th>Entry Velocity [m/s]</th>
<th>0.25</th>
<th>0.5</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall</td>
<td>105</td>
<td>1.516</td>
<td>1.518</td>
</tr>
<tr>
<td>Temperature [°C]</td>
<td>125</td>
<td>1.674</td>
<td>1.696</td>
</tr>
<tr>
<td></td>
<td>140</td>
<td>1.812</td>
<td>1.847</td>
</tr>
</tbody>
</table>

Table 1 Mesh Adaptation Factor $F_{mesh}$

The influence of wall surface roughness on turbocharger fluid dynamics is investigated by Lintz (Lintz, 2017). The subject will be addressed in future studies. It is therefore beyond the scope of the present work.

APPLICATION

Simulation results were validated against a turbocharger with a water-cooling jacket in an aluminium turbine housing and a bearing housing water core. A comprehensive set of experimental data is available for the former. The remainder of this section contains proprietary data i.e. qualitative and non-dimensional data is shown.

CFD results at rated power with and without the SBM show differing wall temperatures on the interface to the water jacket, Figure 12. The wall temperature is normalized by a maximum temperature. The cooler wall temperature with boiling gives an indication of the boiling location, which is further confirmed in Figure 13. It shows the portion
of the wall heat flux attributed solely to nucleate boiling by the SBM. This increased heat flux improves the heat transfer from the wall of the turbine housing into the cooling fluid thus lowering the wall temperature and increasing the temperature of the fluid. The relevant area is situated near the exit of the turbine housing immersed with hot exhaust gas.

The overall wall heat flux portion of the heat balance predicted with the SBM is closer to the experimental data than without, Figure 14. The difference between experiment and simulation is likely to be related to systematic measurement errors such as the closeness of the water temperatures sensors to the wall and the known underprediction of the solver.

Figure 12 Normalized Wall temperature with (left), without boiling model (right)

Figure 13 Normalized Wall heat flux due to nucleate boiling

Figure 14 Overall wall heat flux for experiment and simulation with and without boiling

A further test case of a turbocharger with a water-cooled bearing housing is investigated. These water cores are designed to avoid film boiling within the parameters set by their application. To test the boiling model a bearing housing is considered, which is usually operated with an additional radiator. A hypothetical test case to assess the external cooling requirement is set up: the liquid is uncooled. It is found that nucleate boiling occurs when realistic boundary conditions at rated power are prescribed, Figure 15. The difference in wall temperature with and without boiling is used as an indication. The onset of detrimental film boiling may also be predicted by the SBM but does not occur. I.e. the external cooler is a safety precaution to account for more severe transient boundary conditions only.

Figure 15 Normalized Wall temperature with (left), without boiling model (right)

SUMMARY & CONCLUSIONS
Heat transfer predictions using CFD simulations require physical modelling, robust convergence and sophisticated post-processing. The influence of the computational mesh on the results needs to be quantified. The advanced “Simplified Boiling Model” (SBM) is shown to predict nucleate boiling sufficiently well for industrial use. The computation of total heat transfer coefficients and fluid temperatures are achieved by localization of the thermal boundary layer thickness via the related temperature gradient (DTG) or y-plus value (DYF) methods.

The most prominent further findings are:
• The temperature gradient (DTG) and y-plus (DYF) methods yield almost identical results but the DYF method is more robust.

• Wall Function and LRN computational meshes lead to different levels of wall heat flux compared to experimental data. A mesh adaptation factor is therefore proposed. LRN meshes are recommended for heat transfer prediction without boiling. However, the fine wall resolution of these meshes is not suitable for the SBM model.

• Application and validation of the SBM model within a turbocharger aluminium turbine housing demonstrates its prediction capability. The SBM also becomes a valuable tool in the design of water cores for bearing housings to avoid film boiling. The approach to compute HTCs may be applied to heat transfer problems in general.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CHT</td>
<td>Conjugate Heat Transfer</td>
</tr>
<tr>
<td>DTG</td>
<td>Diabatic Temperature Gradient</td>
</tr>
<tr>
<td>DYF</td>
<td>Diabatic Y-plus Field</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>LRN</td>
<td>Low Reynolds Number</td>
</tr>
<tr>
<td>HTC</td>
<td>Heat Transfer Coefficient</td>
</tr>
<tr>
<td>SBM</td>
<td>Simplified Boiling Model</td>
</tr>
<tr>
<td>(c_{\text{evap}})</td>
<td>Evaporation Constant</td>
</tr>
<tr>
<td>(c_p)</td>
<td>Isobaric Heat Capacity</td>
</tr>
<tr>
<td>(D_\Phi)</td>
<td>Kinematic Diffusivity for the Scalar (\Phi)</td>
</tr>
<tr>
<td>(m)</td>
<td>Mass Transfer Between Phases</td>
</tr>
<tr>
<td>(p)</td>
<td>Pressure</td>
</tr>
<tr>
<td>(Re)</td>
<td>Reynolds Number</td>
</tr>
<tr>
<td>(S_\varphi)</td>
<td>Volumetric Source Term</td>
</tr>
<tr>
<td>(T)</td>
<td>Temperature</td>
</tr>
<tr>
<td>(q)</td>
<td>Heat Flux</td>
</tr>
<tr>
<td>(S_{\text{local}})</td>
<td>Boiling Suppression Factor</td>
</tr>
<tr>
<td>(y)</td>
<td>Wall Distance</td>
</tr>
<tr>
<td>(y^+)</td>
<td>Dimensionless Wall Distance</td>
</tr>
<tr>
<td>(\alpha)</td>
<td>Heat Transfer Coefficient</td>
</tr>
<tr>
<td>(\Delta q_\varphi)</td>
<td>Latent Heat</td>
</tr>
<tr>
<td>(\eta)</td>
<td>Dynamic Viscosity</td>
</tr>
<tr>
<td>(\lambda)</td>
<td>Thermal Conductivity</td>
</tr>
<tr>
<td>(\rho)</td>
<td>Density</td>
</tr>
<tr>
<td>(f_{\text{wall}})</td>
<td>Wall Shear Stress</td>
</tr>
<tr>
<td>(\nu)</td>
<td>Kinematic Viscosity</td>
</tr>
<tr>
<td>(\sigma)</td>
<td>Surface Tension</td>
</tr>
<tr>
<td>(\Phi)</td>
<td>Concentration (Conserved Quantity per unit volume)</td>
</tr>
<tr>
<td>(\varphi)</td>
<td>Conserved Quantity per Unit Mass</td>
</tr>
<tr>
<td>(\varphi)</td>
<td>Volume Fraction</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Subscript</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>l</td>
<td>Liquid</td>
</tr>
</tbody>
</table>

Acknowledgements

The authors wish to gratefully acknowledge the support by Stephan Erhardt, Marc Gugau, Piotr Łuczyński, Patrick Weber and Manfred Wirsum.

REFERENCES


