EULER-EULER SCALE-ADAPTIVE SIMULATION OF A SQUARE CROSS-SECTIONAL BUBBLE COLUMN

T. Ma¹, D. Lucas¹, T. Ziegenhein¹ and J. Fröhlich²

¹Helmholtz-Zentrum Dresden-Rossendorf, Dresden, Germany
²Technische Universität Dresden, Dresden, Germany

tian.ma@hzdr.de

Abstract

In this paper we present detailed Euler-Euler Scale-Adaptive Simulation (SAS) of dispersed bubbly flow in a square cross-sectioned bubble column. The main objective of this study is to investigate the potential of this approach for the prediction of bubbly flows with anisotropic liquid velocity fluctuations, in terms of mean quantities. The set of physical models describing the momentum exchange between the phases was chosen according to previous experiences of the authors. Experimental data and Euler-Euler Large Eddy Simulation (LES) are used for comparison. It was found that the presented modelling combination provides very good agreement with experimental data for the mean flow and liquid velocity fluctuations. The energy spectra obtained from the resolved velocity from Euler-Euler SAS and LES are presented and in comparison with the experimental one.

1. Introduction

Many technical flows situations feature a continuous liquid phase and a dispersed gaseous phase. The turbulence of the liquid phase is an important phenomenon in such multiphase flows. As it has a strong influence on the local distribution of the dispersed phase, including bubble coalescence and breakup.

A bubble column provides a good experimental system for the study of turbulent phenomena in bubbly flows and the development of computational models. In bubble columns a wide range of length and time scales exists on which turbulent mixing takes place. The largest turbulent scales are comparable in size to the characteristic length of the mean flow and depend on the column geometry and boundary conditions. The smaller scales depend on the bubble dynamics and are proportional to the bubble diameter. In bubbly flows, the small scales are responsible for the dissipation of the turbulent kinetic energy as in single-phase flow, but the bubbles can also generate back-scatter, i.e. energy transfer from smaller to larger scales (Dhotre et al. 2013). The combination of both effects can yield an overall enhancement or attenuation of the turbulence intensity.

In the present work several turbulence models are investigated. In CFD simulations of bubble columns, Reynolds-Averaged Navier-Stokes (RANS) models are traditional, and in principle all RANS models can be solved in unsteady mode (URANS). However, these models consider only isotropic turbulence and do not aim at resolving any turbulent motion, even if the grid and time step would be small enough. For this reason the use of classical URANS models to predict the bubbly flows with strong anisotropic velocity fluctuation could be critic.

Scale-Resolving Simulations (SRS) are in general capable to reflect the anisotropy of turbulence. Large Eddy Simulation (LES) offers the opportunity to resolve the large-scale anisotropic turbulence directly and to model the small scales with a Subgrid-Scale (SGS) model. But this approach is too expensive for most industrial flows due to high resolution requirements especially in boundary layers. Scale-Adaptive Simulation (SAS) approach (Menter et al. 2010) is based on an exact transport equation for the turbulent length scale using von Karman length-scale. The information provided by the von Karman length-scale allows the SAS model to adjust to resolved structures in an (unsteady) URANS simulation. So that generally, less computational effort is required compared to LES.

In the present work, Euler-Euler SAS are performed for a square cross-sectioned bubble column (Figure 1). To the best of the author’s knowledge, this is the first use of SAS for this type of flow. First, we discuss the influence of the turbulence, by comparing the experimental data from Deen et al. (2000) and own LES. Second, the effect of modelling bubble-induced turbulence (BIT) with these two approaches is discussed. In the comparison with the experiment, the unresolved turbulent kinetic energy in both SAS and LES will be considered.
liquid phase, respectively. It was found that this turbulence in the dispersed gas phase is of little relevance and is modelled with a molecular viscosity and resolved velocity, 

\[ \frac{\partial}{\partial t} \left( \alpha_i \rho_i u_i \right) + \nabla \cdot (\alpha_i \rho_i \mathbf{u}_i) = -\nabla \cdot (\alpha \mathbf{u} \rho_i \mathbf{u}_i) - \alpha \mathbf{u} \rho_i \mathbf{g} + \mathbf{M}_i - \nabla (\alpha_i \mathbf{r}_i). \]

Here, the lower index \( i \) denotes the different phases, with \( \alpha, \rho, \mu \) and \( \mathbf{u} \) the volume fraction, density, molecular viscosity and resolved velocity, respectively. \( \mathbf{S} \) is the strain rate tensor. The vector \( \mathbf{M} \) represents the sum of all interfacial forces acting between the phases such as drag force, lift force, wall lubrication and turbulent dispersion force. The unresolved stress tensor \( \mathbf{r} \), and all interfacial forces have to be modelled. The applied modelling is discussed below.

### 2.2 Turbulence

In this study, turbulence is treated differently for the two phases. The turbulence in the dispersed gas phase is of little relevance and is modelled with a simple zero equation model \( \mu_i^G = \frac{\mu_i^L}{\mu_i^G} \), where \( \mu_i^G \) and \( \mu_i^L \) are the eddy viscosity of the gas and the liquid phase, respectively. It was found that this model has nearly no influence on the result, because of the low density of the gas and the low volume fraction in this case. For the liquid phase, SAS and LES were used in separate runs.

### 2.2.1 SAS for continuous liquid phase

The liquid velocity \( \mathbf{u}_L \) in (1) and (2) represents the resolved part only. The corresponding unresolved components are:

\[ \mathbf{u}^L = \mathbf{u}_L - \mathbf{u}_L. \]  

where \( \mathbf{u}_L \) is the true velocity of the liquid. The unresolved stress tensor \( \tau_{ij} = \overline{\mathbf{u}_L \mathbf{u}_L} - \overline{\mathbf{u}_L \mathbf{u}_L} \), is modelled by the SAS model of Menter and Egorov (2010). It employs an equation for the turbulent kinetic energy \( K \) and an equation for \( \Phi = \sqrt{\overline{\mathbf{r} \cdot \mathbf{r}}} \), with \( L_L \) the turbulent length scale, reading:

\[ \frac{\partial (\rho K)}{\partial t} + \frac{\partial (\rho \mathbf{u}_L K \mathbf{u}_L)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \alpha \mathbf{S}_{ij} - \frac{\mu_i \partial K}{\partial x_j} \right) - \frac{\partial}{\partial x_j} \left( \alpha \mathbf{S}_{ij} \right) \]

Here, \( L_{\text{vk}} \) is the von Kármán length-scale defined by

\[ L_{\text{vk}} = \kappa \left| \frac{\partial U/\partial y}{\partial^2 U/\partial y^2} \right| \]

The values of the constants are \( \zeta_1 = 0.8, \zeta_2 = 1.47, \zeta_3 = 0.0288, \sigma_B = 2/3, \kappa = 0.41, c_m = 0.09 \). The information provided by the von Kármán length-scale allows the SAS model to treat large turbulent structures in a URANS fashion so that less computational effort is required compared with LES. The \( L_{\text{vk}} \) term can be transformed and implemented into any other scale-defining equation. Here, the Shear Stress Transport model (SST) is used, resulting in an SAS-SST model. Menter and Egorov (2005) feature an additional SAS source term \( F_{\text{sas}} \) in the \( \omega \)-equation resulting from the transformation. The finally used version of this source term reads (Egorov and Menter 2008):

\[ F_{\text{sas}} = \max \left\{ \frac{\rho \kappa \alpha \alpha^2}{\sigma_\phi} \left( \frac{L_{\text{vk}}}{S} \right)^2 - \frac{2 \rho K}{\sigma_\phi} \max \left\{ \frac{1}{\omega^2} \frac{\partial \omega}{\partial x_j} \frac{\partial K}{\partial x_j}; \frac{\partial K}{\partial x_j} \right\} \right\} \]

with \( \zeta_2 = 3.51, c = 2 \) and \( S = \sqrt{2} \sigma_\phi \). The equilibrium eddy viscosity of the SAS-SST is:
\[ \mu_t = \rho \left( \sqrt{\left( \frac{\beta}{C_v} - a \right) / (\kappa \varepsilon^2)} \cdot L_{eq} \right)^2 S. \]  

(8)

with \( \beta = 0.075 \). This formula has a similar structure as the Smagorinsky subgrid scale (SGS) eddy viscosity used for the LES in this study:

\[ \mu_{sgs} = \rho (C_s \Delta)^2 S. \]  

(9)

Here, the model constant \( C_s \) was chosen to be \( C_s = 0.12 \), while the filter width \( \Delta \) was determined by the grid size from \( \Delta = \sqrt[3]{V} \cdot \Omega \). Comparing to LES, SAS resolves the unsteady structure without explicit grid dependency. Instead, a second physical length–scale is introduced to reduce the eddy viscosity compared to URANS (Fröhlich and von Terzi 2008).

### 2.2.2 Bubble-induced turbulence

In the Euler-Euler approach bubbles are treated statistically, i.e. single bubbles are not resolved. The resolved part of the velocity field in SAS and LES produces only the shear-induced turbulence. The influence of bubbles traveling through the liquid on the liquid turbulence has to be modelled. Usually it is assumed that all energy lost by the bubble due to drag is converted to turbulent kinetic energy in the wake of the bubble. For classical two equation RANS models, additional source terms have been developed to describe BIT. Detailed information about the BIT models in RANS can be found in the recent review by Rzehak and Krepper (2013). Such an approach is not suitable for LES because there is no transport equation for \( K \) available. For SAS, up to now, non BIT models are available in literature. Here, we use the common BIT model of Sato et al. (1981). In this model the bubble influence in liquid turbulence is included by an extra term to the SGS turbulent viscosity so that:

\[ \mu_{\text{eff}} = \mu_{L}^{\text{mol}} + \mu_{L}^{\text{sgs}} + \mu_{L}^{\text{pub}}, \]  

(10)

\[ \mu_{L}^{\text{pub}} = C_B \rho_L a_G d_B |u_G - u_L|. \]  

(11)

Here, \( C_B \) is a model constant equal to 0.6, and \( d_B \) represents the bubble diameter. In SAS, the source for unresolved turbulent kinetic energy is proportional to the modified turbulent viscosity \( \mu_{L}^{\text{mod}} \):

\[ P_K = \mu_{L}^{\text{mod}} S^2 = (\mu_{L}^{\text{eff}} + \mu_{L}^{\text{pub}}) S^2. \]  

(12)

In LES, \( \mu_{L}^{\text{pub}} \) is added in the SGS model, without further changes. The comparison of the influence of the Sato model for SAS and LES will be discussed in the section on results below.

### 2.3 Interfacial forces

In the Eulerian two-fluid model the interaction between the bubbles and the liquid phase is modelled through exchange terms in the momentum equation of the liquid and the gas phase. There is still no agreement in the community on the closures to be used. The corresponding interfacial transfer models employed here are listed in Table 1. A complete description of all interfacial transfer models can be found in the so-called baseline model of Helmholtz-Zentrum Dresden-Rossendorf (Rzehak and Krepper 2013).

Table 1: Interfacial forces models employed for both cases in the present work.

<table>
<thead>
<tr>
<th>Drag force</th>
<th>Ishii and Zuber (1979)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lift force</td>
<td>Tomiyama et al. (2002)</td>
</tr>
<tr>
<td>Wall force</td>
<td>Hosokawa et al. (2002)</td>
</tr>
<tr>
<td>Virtual mass</td>
<td>( C_{VM} = 0.5 )</td>
</tr>
</tbody>
</table>

**Turbulent Dispersion Force**

The turbulent dispersion force represents the bubble dispersion caused by the turbulent fluctuations of the liquid velocity. In URANS simulations, it has to be modelled; because these turbulent fluctuations are not simulated. In LES, however, the resolved part of the turbulent dispersion is explicitly calculated, and the unresolved part has little influence on bubble dispersion if the bubble size is on the scale of the filter size (Nicenko et al. 2008). In contrast, using SAS the unresolved part is often not negligible, and larger than in LES. Here, the model from Burns et al. (2004) derived by Favre averaging of the drag force is used:

\[ F_{\text{disp}} = - \frac{3}{4} C_D \frac{a_G}{d_B} |u_G - u_L| \mu_L^{\text{eff}} \left( \frac{1}{\sigma_{TP}} \frac{1}{a_G} \right) \nabla \bar{u}_G. \]  

(13)

with \( C_D \) the drag coefficient according to the correlation by Ishii and Zuber (1979) and \( \sigma_{TP} \), the turbulent Schmidt number taken to be 0.9.

### 3. Experimental data and simulation setup

**Experimental data**

The simulations were carried out for an air/water bubble column at ambient pressure (Deen et al. 2001), schematically shown in Figure 1. The bubble column has a square cross-section of 0.15 m \( \times \) 0.15 m and is filled with distilled water up to a height of 0.45 m. A distributor plate containing 49 needles with an inner diameter of 1 mm was placed in the centre of the bottom with a square section of \( A_{in} = 0.03 \text{ m} \times 0.03 \text{ m} \). Measurements were performed for the gas superficial velocity of 4.9 mm/s and took place 0.25 m above the distributor.
plate \((y = 0.25 \text{ m})\) in the centre plane \((z = 0.075 \text{ m})\). More details are provided in the cited reference.

**Simulation Details**

The gas inlet is defined as distributed source in the central area \(0.03 \times 0.03 \text{ m}^2\) at the bottom of the domain, representing the experimental needle setup. The inlet velocity is equal to the gas volume flow divided by the total inlet surface. At the walls, a no slip boundary condition is applied for the continuous phase and a free slip condition for the disperse phase. At the top of the column a degassing boundary is imposed, which means a slip condition for the continuous phase and an outlet for the dispersed phase. For the spatial discretization, a central difference scheme is employed, and a second-order backward Euler scheme is used in time. The simulations were carried out using the time steps \(\Delta t = 0.0025 \cdots 0.005\) to satisfy \(\text{CFL} < 1\). The simulations were run for 200 s.

**Grid Requirement**

The rectangular bubble column was discretized with uniform cubic cells of \(\Delta x = \Delta y = \Delta z = 5 \text{ mm}\), resulting in 81000 cells, overall. The spatial resolution hence is in the order of the bubble size \((\Delta x = d_p)\). The Euler-Euler method is a volume-averaging approach. Hence, there is no strict requirement, like for most of the bubble tracking methods, that the grid size should be larger than the (physical) bubble diameter when modelling interfacial forces (Sungkorn et al. 2011). But if the bubble diameter is larger than the grid size, the bubble’s interfacial detail becomes important and should be resolved (Niceno et al. 2009). In the present work, the grid size was chosen as a compromise between sufficiently fine resolution to capture the most energetic eddies, and sufficiently coarse grid to consider the limit of Euler-Euler approach. The grid size used in the present work was chosen according to the mesh studies of Niceno et al. (2008) and Sungkorn et al. (2011) to supply the best prediction. The simulations were conducted with the software Ansys CFX 14.5.

**4. Results**

**4.1 Instantaneous results**

Snapshots of the isosurfaces representing a constant void fraction and the void fraction distribution in the centre plane with SAS and LES are shown in Figure 2. On average, the bubbles reach the top of the column after about 1.2 s. The left graphs show the isosurfaces of gas volume fraction equal to 0.05 obtained with SAS and LES. These isosurfaces give good illustrations of the dynamics of the bubble plume and look very similar to the numerical results presented in Niceno et al. (2009). For LES, the plume is more distorted than in case of SAS. This is caused by the smaller effective viscosity compared to SAS, leading to a better resolution of small fluctuations. The right graphs depict the corresponding void fraction in the central plane, showing a difference between the results for the two turbulence models. In the results of SAS, due to the application of turbulent dispersion model, the gas volume fraction is distributed somewhat more homogeneously over the cross section. It was confirmed by an own simulation, that using SAS without turbulent dispersion model yields a bubble plume rising up to the top of the column without much transversal spreading. This is in contrast to the experiment, where the bubble plume moves around in the bubble column in a random fashion (Deen et al. 2001).

![Figure 2: Instantaneous data from SAS and LES at 70 s. The left two graphs show instantaneous isosurfaces of constant void fraction \((\alpha = 0.05)\). The right two graphs show the instantaneous void fraction in the centre plane.](image-url)
4.2 Time-averaged results

In this section, the simulation results averaged from 20 to 200 s are quantitatively compared with the experiment. The experimental data were taken at the centerline of the horizontal plane at height 25 cm, as indicated in Figure 1. Time averaged vertical liquid and gas velocity, vertical and lateral liquid velocity fluctuation, gas void fraction and effective viscosity are presented for both cases. Figure 3 and 4 show profiles of the averaged vertical liquid and gas velocity. Satisfying quantitative agreement with the experiment is obtained for both simulations. The results of SAS have a bit lower values compared to the LES in both profiles. The liquid velocity profile is somewhat underpredicted with SAS, but well reproduced using LES. On the other hand, the gas velocity agrees better using SAS compared to the LES. This effect can be attributed to the drag force modelling and should be subject of further investigations. In both simulations the relative velocity between liquid and gas is about 0.2 m/s, i.e. it is nearly the same, resulting from the use of the same drag force model. The vertical liquid velocity changes its sign about 0.02 m away from wall, which is in agreement with the experimental data. The downflow in the near-wall region is caused by liquid mass balance in the bubble column.

Figure 5 and 6 present a comparison of the vertical and lateral fluctuating liquid velocity. In the previous works by Deen et al. (2001) and Dohrte et al. (2008), only the resolved velocity fluctuations were provided, while the unresolved part was neglected. In the present SAS and LES, the total velocity fluctuation is considered, summing the resolved part \( \bar{u}' \) and the unresolved part \( u' \) as mentioned in Section 2.2 above. For the resolved part, \( \bar{u}' = [\bar{u} - \langle \bar{u} \rangle]^2 \) is computed using the difference between the instantaneous resolved velocity \( \bar{u} \) and the time average of the resolved velocity, \( \langle \bar{u} \rangle \). Modelling the unresolved part is different for SAS and LES. SAS has a transport equation for the turbulent kinetic energy. Assuming isotropy of turbulence leads to \( \nu' \frac{\partial^2}{\partial x^2} \bar{u}' \). For LES, only the anisotropic part of the SGS stress tensor is considered in the Smargorinsky model, so that the information about the isotropic part, \( \nu_{kk} = 2K \), is lost. Here, the following estimate for \( K_{sgs} \) is used (Menter 2013):

\[
K_{sgs} = \sqrt{\frac{\nu_{sgs} \varepsilon_{sgs}}{C_\mu}}, \quad \varepsilon_{sgs} = \nu_{sgs} S^2, \quad (14)
\]

with \( \varepsilon_{sgs} \) the SGS dissipation. The constant \( C_\mu \) is taken from the \( k-\varepsilon \) model.

The qualitative comparison of the fluctuations of the vertical liquid velocity shown in Figure 5 is
encouraging for both, SAS and LES. However, the
twin-peaked shape seen in the experiments is only
reproduced by the SAS. The unresolved vertical
velocity fluctuation with the SAS is higher than that
with the LES, due to the higher effective viscosity
of the SAS (see Figure 8 below). This effect is
expected and matches the basic concept of
Corresponding hybrid LES/RANS methods (here
SAS) as discussed by Fröhlich and von Terzi
(2008). The fluctuating lateral liquid velocity
shown in Figure 6 is not predicted as well as the
one of the vertical component. Quantitatively, both
SAS and LES overpredict the magnitude to some
extent. This overprediction is caused by the high
ratio of the modelled lateral fluctuations to the total
lateral fluctuations (about 35% in SAS and 16% in
LES). Compared to the vertical direction, the
resolved fluctuations in lateral direction are much
smaller. But determining the unresolved parts
isotropic turbulence is assumed with both
turbulence models and provides the same values as
for the vertical direction.

Effect of using Sato Model

Previous works from Deen et al. (2001), Dohrte et
al. (2008) and Niceno et al. (2008) mentioned that
there was no significant change in the predictions
using LES with and without the BIT model of Sato.
This is not confirmed by our results.

Figure 7: Vertical liquid velocity obtained from
SAS, LES and experiments, with and without Sato
model.

In Figure 7, the mean vertical liquid velocity
profiles with and without Sato BIT model are
shown for both turbulence models. There is a more
pronounced peak in the centre with both
approaches, when the model is included, clearly
overpredicting the experimental data. This can be
attributed to the fact that, as shown in Figure 8,
with the additional viscosity caused by the Sato
BIT model in (11), the effective viscosity increases
significantly in both SAS and LES, which in turn
damps the bubble plume dynamics. A behaviour
similar to Figure 7, with an even higher centre peak
in the vertical liquid velocity profile was obtained
by Zhang et al. (2006). He performed LES with an
increased value of \( C_s = 0.2 \) in the Smagorinsky
SGS model instead of \( C_s = 0.12 \). The application
of the Sato model in the LES has a similar effect as
increasing \( C_s \), since it enhances the SGS viscosity.

Figure 8: Liquid phase effective viscosity obtained
from SAS and LES, with and without Sato model.

Figure 9: Gas void fraction obtained from SAS,
LES, with and without Sato model.

Figure 9 presents the comparison of the void
fraction with and without the Sato model. Using the
model in both SAS and LES, a more pronounced
peak in the centre is obtained, compared to a flatter
profile without the model. The larger gas volume
fraction in this region also increases the liquid
velocity in the centre and thus is a second reason
for the steeper velocity profiles. Unfortunately, a
comparison with the experimental data is not
possible, because no void fraction profiles were
measured.

In the earlier results of Deen et al. (2001), Dohrte et
al. (2008) and Niceno et al. (2008) who also used
the Sato model with LES, an increase of vertical
velocity in the centre was not observed as in the
present work. One reason could be the fact that they
used another lift coefficient \( C_l = 0.5 \). In the present
study, the lift force correlation of Tomiyama is us ed
with an effective lift coefficient \( C_l \approx 0.28 \). As
reported by Lucas and Tomiyama (2011), the lift
force may have a significant effect on gas plume
transverse spreading.
4.3 Analysis of spectra

Figures 10 and 11 show time series of the resolved vertical liquid velocity at the centre of the measurement line (Figure 1) for SAS and LES without BIT model. The period of 50 s presented in these figures corresponds to 189 sample points for the experimental data and 500 sample points for the simulations. Both, SAS and LES reflect the transient behaviour observed in the experiment of Deen et al. (2001) in terms of the amplitude of the fluctuations. The differences observed in the signals are caused by the different sampling rates, so that an investigation of the spectra is helpful.

Figure 10: Time history of the liquid vertical velocity obtained with SAS at the centre of measurement line, compared to experimental data from Deen et al. (2001).

Figure 11: Time history of the liquid vertical velocity obtained with LES at the centre of measurement line, compared to experimental data from Deen et al. (2001).

Figure 12: Power spectrum density of vertical liquid velocity taken at the centre of measurement line for SAS and LES in comparison to the result obtained from original experimental time signals.

The power spectrum densities (PSD) of SAS and LES without BIT model obtained with 18000 sample points are shown in Figure 12. The data are also extracted from the centre of the measurement line and considers the time from 20 to 200 s. A fast Fourier transform of the vertical liquid velocity signal using the Welch method (Welch 1967) with non overlapping sections and a Hanning window was performed. Because of the low sample frequency, the experimental spectrum can only be determined up to about 4 Hz.

Both spectra with SAS and LES exhibit a broad range of frequencies and slopes of about -5/3 in the inertial subrange, like single phase turbulent flow. Both predictions are in good agreement with the experiment in the low frequency region. Hence the large-scale vortical magnitudes are well reflected in the simulations. Compared to LES, the spectrum of SAS is more damped caused by the larger eddy viscosity. This is a similar effect as increasing the Samgorinsky constant $C_S$ in a LES, which is discussed by Fröhlich (2006).

5. Conclusions

The simulation results obtained with SAS for a bubble plume show good agreement with LES results and experiment, both for the instantaneous bubble plume shape and for the mean-flow properties. The turbulence quantities determined with SAS are similar in quality as those obtained from LES in this case. Especially the trend of the fluctuations in the mean flow direction is well captured. SAS hence might be a promising option to be used with the Euler-Euler approach for the interfacial momentum exchange to predict bubbly flows.

The Sato BIT model was investigated in SAS and LES. It is found, that the increased effective viscosity resulting from this model damps the bubble plume dynamics. This causes a more pronounced peak in the centre of the liquid velocity profile compared to experimental one. Such an effect of the Sato BIT model was not found by Ma et al. (2014) and Ziegenhein et al. (2014) for another bubble column with a homogeneous distributed gas inlet at the bottom. One reason could be the flatter gas volume fraction profile in their simulated bubble column. The impact of the increased effective viscosity caused by the Sato model for this reason is much higher in the present case.

Further investigations of the relationship between resolved and non-resolved turbulence and interfacial force modelling will be conducted in the future for different bubbly flow configurations.

Acknowledgments

The authors would like to thank N. Deen for making the experimental time signals available for this work.
This work is carried out in the frame of a current research project funded by the German Federal Ministry of Economic Affairs and Energy, project number 150 1411.

References


F.R. Menter, private communication, 2013.


