

Development of a Numerical Simulation Tool for the Cooling of Batteries

Thorsten J. Möller, Ruben Stahlbaum, Choopong Sereechooenkij and Rolf Radespiel

Technische Universität Braunschweig, Institut für Strömungsmechanik

Copyright © 2008 SAE International

ABSTRACT

The objective of the present work is to establish a numerical simulation methodology for the pressure loss for air cooling of batteries that can be used in the industrial design process. The 2D numerical flow calculations focus on the flow in the entire test rig. To validate the numerical results an existing test rig was used comprising a pack of prismatic batteries. The flow field is analysed using Particle Image Velocimetry and pressure measurements. The results display a detailed account of the various flow phenomena and their scales that contribute to the complex flow problem of air cooled batteries. They offer a methodology to simulate the pressure loss and highlight the technical challenges to obtain homogeneous cooling.

INTRODUCTION

In industrial design processes it is helpful and cost reducing to have a validated numerical simulation methodology at hand. Using such a methodology enables the developer to judge the performance of a new concept. Previous works mainly focused on the experimental evaluation of the thermal behaviour of existing battery packs [1] and the module design [2], but less is published regarding to a validated numerical model, which is the subject of the present work.

The flow path of the investigated battery pack is similar to a z-type manifold like is also common in fuel cells [3]. In [4] the pressure loss of fuel cells is investigated by means of a numerical simulation. Due to the investigated geometry it was possible to compute the flow field and the resulting pressure loss inside the fuel cell without any additional modelling. First investigations presented in [5] identified different complex flow phenomena like backward facing step [6, 7], free jet in crossflow (JICF) [8, 9], flow through the cooling gap with a complex gap geometry. In this first phase of the project [5] it has become obvious a 3D numerical simulation taking into

account full detail is very costly. To reduce the computer costs the concept of porous medium [10] could be applied to model the pressure loss of flow through the cooling gaps. This approach is used in [10] to model the pressure loss in a z-type manifold in a fuel cell [11].

In the first phase of the project [5] also a test rig was designed and set up built up, to experimentally simulate the flow field around a pack of prismatic batteries to obtain data which can be used for the validation of numerical results. Well defined and reproducible boundary conditions are needed for simulations, so the quality of the inflow has been documented in detail. The flow can be divided into parts. One is the flow towards and away from the cooling gaps and the other is the flow through the cooling gaps itself. This is the so called z-type flow path [12]. Pressure loss due to the flow through cooling gaps is an essential parameter for the efficiency of cooling.

Further experimental investigations using Particle Image Velocimetry (PIV) were conducted to obtain detailed data for the validation of 2D numerical simulations applying different turbulence models and discretisation schemes. Furthermore a comparison between numerical and experimental results of the flow field outside the gap was conducted. Finally pressure measurements were performed to determine the pressure loss for all gaps in the battery pack. This data was also compared to numerical results.

EXPERIMENTAL EQUIPMENT AND MEASUREMENT TECHNIQUES

TEST RIG

To obtain reliable data for the validation of numerical data a sound experimental environment is necessary. An experimental rig of variable size was therefore designed and constructed. It is displayed in figure 1.

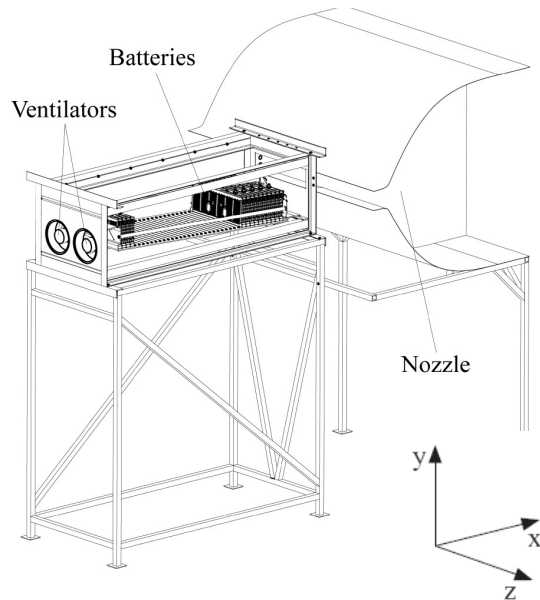


Figure 1: Sketch of the test rig

The rig includes a nozzle, a test section and a propulsion unit. Geometric variability is given in all three spatial directions. The nozzle is designed following the method of Boerger [13] for a 2D layout. This method provides the nozzle contour so that the velocity at the outlet is uniform. Hence the inflow boundary condition at the inlet of the test section is well defined. The nozzle is also variable in all three spatial directions. The present inlet area measures $270 \times 70 \text{ mm}^2$. The test section is constructed of a frame of L-profiles. Due to this, variability of all three spatial directions is given. The back board consists of a plate of steel. Bores in the back board allow the electric power supply of the propulsion and the export of electrical and other measurement signals. The top, front and bottom are made of glass in order to ensure access with optical measurement techniques, e.g. Particle Image Velocimetry (PIV). In the centre of this housing the battery pack is installed. The propulsion is performed by two axial ventilators. Further details of the test rig are given in [5].

THE BATTERY PACK DISCRIPTION

The batteries used are prismatic NiMH modules. This battery type is used in operating hybrid electric vehicles. Each module, as displayed in figure 2, contains six 1.2 V cells that are connected in series.

Thus, the nominal voltage is 7.2 V at a capacity of 6.5 Ah. The dimensions are 275 mm (L) x 106 mm (H) x 19.6 mm (W). The weight of a module is 1.04 kg. On the top there is a hollow which allows measurement of the temperature of the electrodes. Next to this hollow, there is a cylindrical hydrogen vent which minimizes the risk of rupture due to strong heating.

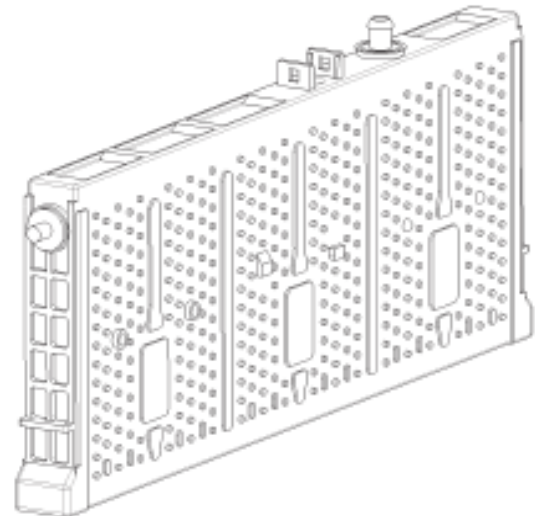


Figure 2: View of the battery module

The side surfaces form gaps that allow air cooling. These channels have a dimension of $106 \times 2 \text{ mm}$ each. On the surfaces there are 1 mm long cylinders with a diameter of 1 mm and some rectangular-like shaped structures. The cylinders are spacers and keep the 2 mm wide cooling gap in shape, in case of thermal expansion of the batteries. Terminals on each side provide clean connections. The battery pack consists of 40 of the previously described prismatic modules. The batteries are connected in series which results in a nominal voltage of 288 V. The modules are stacked side by side and compressed together by the use of fastening bolts. The arrangement is then screwed onto a thin sheet of metal to keep it in place using bolts. The metal sheet is tilted towards the horizontal plane by about 3.5° and fastened to the structure of the test section. The gaps are numbered beginning with zero between the module next to the ventilators and the sidewise structure. Gap number 40 is formed by the module next to the nozzle and the sidewise structure.

DUMMY MODULE AND PRESSURE MEASUREMENTS

The investigation of the flow field inside the cooling gaps cannot be performed by means of optical measurement methods due to the small size of the cooling gap and the material the batteries are made of. Therefore, a dummy of one battery module was constructed using Stereo Lithography. Figure 3 shows the locations of the applied pressure tabs. The pressure tabs were subsequently drilled into the surface to a depth of 5 mm. A second drilling from the sides then leads the pressure line to a flexible tube that was connected to an multiple tube manometer. To determine the small pressure of around 100Pa inside the cooling gaps an inclined tube manometer has been applied with a precision of $\pm 0.6 \text{ Pa}$.

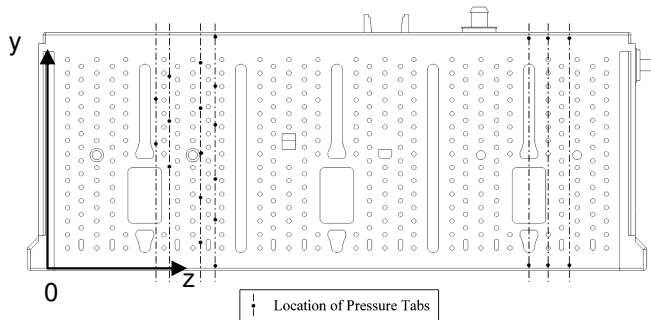


Figure 3: Front of the dummy module

PIV MEASUREMENT TECHNIQUE

Particle Image Velocimetry (PIV) [14] was used for flow field measurements. The PIV images were recorded using one Peltier cooled CCD camera. It comprises of a 1280 x 1024 pixel CCD chip. A 52 mm lens was fitted to the camera. Tracer particles were produced using a 12 hole nozzle atomizer. The fluid used to produce the tracer particles was Diethylhexasebacate (DEHS). The particles have a mean diameter of 0.9 μm . The illumination of the seeding was done by a light sheet using a pair of frequency doubled 115 mJ Nd:YAG LASERs. The experimental setup is displayed in figure 4. The light sheet was directed tangential to the flow direction and perpendicular to the horizontal. For each measurement a double frame picture was taken during a small time interval. The motion of the fluid is shown by the displacement of the tracer particles. An image analysis including detection of the particles in the two pictures and an autocorrelation analysis puts out the velocity field. The mean was computed of 1000 analysed double frame pictures.

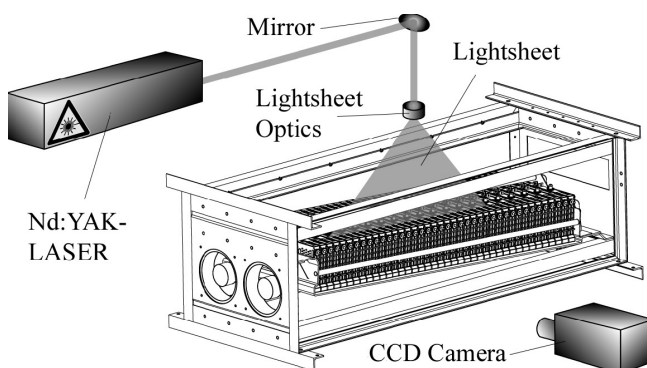


Figure 4: PIV measurement setup

CASES INVESTIGATED AND COMPUTATIONAL DETAILS

The numerical investigations which were performed used the commercial CFD code StarCD [15]. The code is based on the finite volume method that solves the discrete Navier-Stokes equations. The discrete equations are solved in a segregated manner with the SIMPLE algorithm (Semi-Implicit Method for Pressure-Linked). In the SIMPLE algorithm the continuity equation

is converted into a discrete Poisson equation for pressure. The differential equations are solved implicitly in a sequence starting with the pressure equation. This is called predictor stage. In the corrector stage the momentum equations and the pressure correction equation are solved. The model equations of the turbulence models are solved after the update of both velocity and pressure components. Within this loop the linearised equations for each variable are treated using a linear system solver. StarCD uses a preconditioned conjugate gradient solver to solve the implicit system of equations. Several higher order schemes for spatial discretisation are available and a second order scheme was used. The properties of the fluid were the same in all simulations. The density was set to $\rho = 1.205 \text{ kg/m}^3$ and a value of $1.81 \cdot 10^{-5} \text{ kg/(ms)}$ was applied for the molecular viscosity μ . The fluid was considered to be incompressible.

In most of the technical flow fields, as in the present, turbulence occurs. Turbulence can be interpreted as fluctuations that are superposed to a mean flow. Introducing this concept into the Navier-Stokes equations one gets the Reynolds Averaged Navier-Stokes equations (RANS). In these equations additional variables occur in the stress tensor. Turbulence behaves as additional stresses, the Reynolds stresses. Because of these additional variables the system of equations is not closed any more. To close it one needs turbulence models.

The Reynolds stress tensor may be approximated by the Boussinesq approximation [16], in which the Reynolds stresses are thought to be similar to the molecular stresses. This leads to a class of turbulence models. These are called two equations turbulence models based on transport equations for a velocity scale and a length scale of turbulence that can be combined to compute a turbulent viscosity. In the present work only models out of this class were used.

Standard k- ϵ model

Using the Boussinesq approximation it is possible to derive one transport equation for the turbulent kinetic energy, k , and the dissipation, ϵ . This turbulence model predicts Reynolds stresses too large in flow fields containing strong pressure gradients. The influence of transport terms is not sufficiently large. Thus flow separations can not be predicted correctly. Further there appears a large production of turbulence around stagnation points. Next to walls the turbulent length scale is calculated too large so that this model fails here. Despite of all these limits the standard k- ϵ model is widely used in the industry because it is easy to handle and the numerical expense is relatively low.

Quadratic k- ϵ model

Many real flow fields are anisotropic. Non linear models cater for this defect by adopting non linear relationships between the Reynolds stresses and the rate of strain.