A combined experimental and simulation study of fluid-particle heat transfer in dense arrays of stationary particles

Citation for published version (APA):

DOI:
10.1016/j.ces.2016.04.022

Document status and date:
Published: 21/09/2017

Document Version:
Typeset version in publisher's lay-out, without final page, issue and volume numbers

Please check the document version of this publication:
• A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:
www.tue.nl/taverne

Take down policy
If you believe that this document breaches copyright please contact us at:
openaccess@tue.nl
providing details and we will investigate your claim.

Download date: 23. Sep. 2023
A combined experimental and simulation study of fluid-particle heat transfer in dense arrays of stationary particles

K.A. Buist, B.J.G.H. Backx, N.G. Deen, J.A.M. Kuipers

Multiphase Reactors Group, Department of Chemical Engineering & Chemistry, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

Multiphase and Reactive Flows, Department of Mechanical Engineering, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

HIGHLIGHTS

- Comparison of heat transfer experiments and simulations of well-defined problems.
- Heat transfer probe by reconfiguration of Constant Temperature Anemometer.
- Well-defined arrays of particles, using abacus-like structure.

ARTICLE INFO

Article history:
Received 22 January 2016
Received in revised form
5 April 2016
Accepted 10 April 2016

Keywords:
Heat transfer
Constant Temperature Anemometry
Direct Numerical Simulation
Immersed Boundary method

ABSTRACT

A novel experimental technique is introduced to study fluid-particle heat transfer in dense arrays of stationary particle. First a Constant Temperature Anemometer is reconfigured to a heat transfer probe. The thermal driving force is defined as the difference between the probe temperature and the initial fluid temperature and well known in our system. Second an abacus-like structure is employed to accurately control the solids volume fraction. The solids fraction was varied between 0 and 0.6 with increments of 0.1. The Reynolds number varied between 0 and 800. This combination of approaches allows for a very well-defined system, that can be studied both experimentally and numerically, and as such can serve as a validation of heat transfer studies with Direct Numerical Simulations, in fluid-particle systems. A single particle in unbounded flow, the effect of inter-particle distance and shielding effects for an inline array of three spheres as well as semi-structured arrays of particles are studied.

© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

In order to improve the design and control of gas–solid contactors, like packed and fluidized beds, detailed understanding of hydrodynamics, mass and heat transfer is needed. Many studies have been performed to obtain empirical correlations for fluid-particle mass and heat transfer, Gunn (1978). Most of these experimental studies have been performed over a very specific range of operating conditions. Often effects of wall channelling and or axial dispersion have been neglected (Wakao and Kagei, 1982), leading to a large scatter in the obtained fluid-particle heat transfer rates as seen in Fig. 1 in terms of the Nusselt number.

Several empirical correlations have been proposed for the different situations involving non-isothermal gas-particle flows; single particle in unbounded flow, particles in a packed bed or fluidized beds. The following correlations have been used for reference purposes in this work:

The Ranz–Marshall (Ranz and Marshall, 1952) correlation describes the fluid-particle heat transfer of a single particle in unbounded flow:

$$ Nu = 2 + 0.6 \left( Re^{0.6} Pr^{0.33} \right) \quad 1 < Re < 10^4, \quad 0.6 < Pr < 380 $$

(1)

The Ranz (Ranz, 1952) equation is an adjustment to the above equation that describes the fluid-particle heat transfer in a packed bed:

$$ Nu = 2 + 1.8 \left( Re^{0.6} Pr^{0.33} \right) \quad 1 < Re < 10^4, \quad 0.6 < Pr < 380 $$

(2)

Finally the Gunn correlation accounts for differences in solids fraction ranging from a densely packed bed to a single unbounded
On the other hand Direct Numerical Simulations (DNS) have been used to obtain more insight into the complex fluid-particle heat transfer phenomena in dense gas-particle flows. In these systems the heat transfer is strongly dependent on the fluid flow patterns and the structural features. These simulations allow for modelling of the actual packing geometry and consequently allow to directly relate fluid-particle heat transfer to structural features and fluid flow. It is very difficult to properly validate the DNS, because most information is restricted to limiting and extreme cases. Direct comparison of Nusselt numbers under varying solids fractions and a full range of flow regimes has up till now not been possible, mostly because of the need of a well-defined driving force (Rexwinkel et al., 1997), as well as the need for a well-defined solids volume fraction.

DNS is limited by the domain size that can be treated in reasonable computation times. Therefore DNS is often used in a multi-scale modelling approach (Van der Hoef et al., 2008), where the results of the DNS are used to derive closure laws required in more coarse grained techniques such as Discrete Particle Modelling (DPM) and Two Fluid Modelling (TFM). The latter two techniques allow the study of larger systems.

Most studies addressing validation of DNS data are based on comparison with analytical solutions in limiting cases or comparison with results from empirical correlations. An experimental approach that allows us to control the solids structure and the thermal driving force is therefore needed. Preferably such a technique is capable of changing or controlling the solids fraction in a precise manner. For instance Happel and Epstein (1954) made an abacus like structure to study pressure drop of structures with varying solids fraction. Mankad et al. (1997) used a similar structure to study heat transfer, using a 1.5 cm hollow copper sphere with a resistor to heat the sphere, using a constant power input for the sphere. Yang et al. (2012) used a similar technique to study the heat transfer of structured packed beds of particles where particles are strung in an BCC and FCC structure where the temperature of a couple of particles is measured with an embedded thermocouple. Experiments in fluidized beds can also be performed by addition of a hot freely moving particle to the flow (Parmar and Hayhurst, 2002; Hayhurst and Parmar, 2002). Changes in magnetic properties due to temperature changes have also been used to study heat transfer in fluidized beds (Turton et al., 1989).

In this paper a similar technique as Mankad et al. (1997) will be used. However, in our work an existing piece of equipment is used, i.e. a spherical Constant Temperature Anemometer (CTA) probe. An anemometer is normally used to study the fluid velocity, by calibrating the heat loss of a probe inserted in a flowing fluid, following Kings law (King, 1914). However, instead of coupling the voltage of the Wheatstone bridge to the fluid velocity it is also possible to convert it to a heat loss of the probe. This form of heat loss is called Joule heating and can almost entirely be attributed to the convective heat loss of the probe.

The outline of this paper is as follows: First the methods are described, starting with the Constant Temperature Anemometer, then the Immersed Boundary method to perform DNS is shortly described. Second the experimental setup is described. In the Results section three different types of experiments are reported: the heat transfer of a single particle in unbounded flow is used as a benchmark, following the work of Ranz and Marshall (1952). Furthermore, the effect of shielding/enhancement in fluid-particle heat transfer for an inline array of three particles as well as the effect of inter-particle distance is studied, and finally the effect of solids volume fraction and fluid velocity (particle Reynolds number) is studied for stationary arrays of particles over a wide range of solids volume fractions and particle Reynolds numbers.

### 2. Methods

#### 2.1. Constant Temperature Anemometry (CTA)

Constant Temperature Anemometry (CTA) finds its origin in the work of King (1914). A thin wire or film is placed in a flow and kept

---

**Nomenclature**

<table>
<thead>
<tr>
<th>Roman symbols</th>
<th></th>
<th>Greek symbols</th>
</tr>
</thead>
<tbody>
<tr>
<td>( R )</td>
<td>resistance ( \Omega )</td>
<td>( \alpha )</td>
</tr>
<tr>
<td>( V )</td>
<td>voltage ( V )</td>
<td>( \alpha_f )</td>
</tr>
<tr>
<td>( T )</td>
<td>temperature ( K )</td>
<td>( \rho )</td>
</tr>
<tr>
<td>( Q )</td>
<td>heat loss ( W )</td>
<td>( \mu )</td>
</tr>
<tr>
<td>( Nu )</td>
<td>Nusselt number</td>
<td>( \varepsilon )</td>
</tr>
<tr>
<td>( Re )</td>
<td>Reynolds number</td>
<td>( \sigma )</td>
</tr>
<tr>
<td>( P )</td>
<td>pressure ( Pa )</td>
<td></td>
</tr>
<tr>
<td>( Pr )</td>
<td>Prandtl number</td>
<td></td>
</tr>
<tr>
<td>( Ru )</td>
<td>Rayleigh number</td>
<td></td>
</tr>
<tr>
<td>( A )</td>
<td>surface area ( m^2 )</td>
<td></td>
</tr>
<tr>
<td>( C_p )</td>
<td>heat capacity ( J/kg K )</td>
<td></td>
</tr>
<tr>
<td>( L )</td>
<td>length ( m )</td>
<td></td>
</tr>
<tr>
<td>( d )</td>
<td>diameter ( m )</td>
<td></td>
</tr>
<tr>
<td>( f )</td>
<td>momentum source term ( N/m^3 )</td>
<td></td>
</tr>
<tr>
<td>( h )</td>
<td>heat transfer coefficient ( W/m^2 K )</td>
<td></td>
</tr>
<tr>
<td>( k )</td>
<td>thermal conductivity ( W/m K )</td>
<td></td>
</tr>
<tr>
<td>( r )</td>
<td>radius ( m )</td>
<td></td>
</tr>
<tr>
<td>( u )</td>
<td>fluid velocity ( m/s )</td>
<td></td>
</tr>
<tr>
<td>( q )</td>
<td>volumetric heat source ( W/m^3 )</td>
<td></td>
</tr>
<tr>
<td>( t )</td>
<td>time ( s )</td>
<td></td>
</tr>
</tbody>
</table>
at a constant temperature while the required power is measured. The temperature of the probe is related to the resistance of the probe where the resistance is maintained by adjusting the voltage over the probe, which depends on the fluid velocity. The voltage of the probe is calibrated to the fluid velocity. CTA-probes are used as a means to measure the fluid velocity, and because of its fast response they are often used to study turbulent flows. The working principle of CTA is based on the relation between the fluid velocity and the heat loss of the probe due to convection. The power loss of the probe is directly related to the convective heat loss of the probe because of Joule heating. In this work we use an omni-directional probe by Dantec that consists of a 3.2 mm sphere. Fig. 2 shows the probe tip, Dantec type 55R49. The probe consists of one resistance out of four legs of a Wheatstone bridge, see Fig. 2.

\[ R_p = R_{20} + \alpha_{20} R_{20} (T_p - T_{amb}) \]  

When the two parallel resistor legs are balanced, no voltage will be measured in the bridge. This is achieved by adjusting the resistance \( R_3 \) at the exact same ratio as the top resistances \( R_1 \) over \( R_2 \), so the voltage at point \( D \) (Eq. (5)) will equal that of \( B \) (Eq. (6)):

\[ V_D = \frac{R_p}{R_1 + R_2} \]  

\[ V_B = \frac{R_3}{R_2 + R_3} \]  

\[ R_3 \text{ follows by taking the ratio of the two top resistances and the sum of all the resistances in the probe-leg, i.e. the resistance of the cables, leads and support are taken into account:} \]

\[ R_3 = \frac{R_1}{R_2} (R_p + R_1 + R_3 + R_c) \]  

Table 1 shows the relevant parameters for the probe and the set resistances. The voltage over the probe is a measure for the total heat loss of the probe:

\[ V_p = V_{top} \left( \frac{R_p}{R_1 + R_3 + R_2 + R_c} \right) \]  

The heat loss is then defined as:

\[ Q_p = \frac{V_p^2}{R_p} \]
from which the Nusselt number can be obtained as follows:

\[ \text{Nu} = \frac{Q_p d_p}{\alpha_p (T_p - T_f) k_f} \]  

(10)

2.2. Direct Numerical Simulations

For the Direct Numerical Simulations (DNS) an Immersed Boundary method is used as illustrated in Fig. 3. The computational domain is divided into control volumes using a structured uniform mesh whereas the spherical particles are represented by Lagrangian marker points evenly distributed among the surface of the sphere.

The fluid dynamics and heat transport are described by the conservation equations for mass, momentum and thermal energy:

\[ \nabla \cdot \bar{\rho} \bar{u} = 0 \]  

(11)

\[ \frac{\partial \bar{\rho} u}{\partial t} + \bar{\rho} u \cdot \nabla \bar{u} = -\nabla P + \mu \nabla^2 \bar{u} + f \]  

(12)

\[ \frac{\partial T_f}{\partial t} + \bar{u} \cdot \nabla T_f = \alpha \nabla^2 T_f + \frac{q}{\rho C_p} \]  

(13)

where \( f \) and \( q \) represent the momentum and heat source terms. These terms are used to respectively enforce the no-slip boundary condition for the flow and the Dirichlet boundary condition for the temperature. The numerical method utilizes a staggered computational grid, for more details the interested reader is referred to the work of Tavassoli et al. (2013). The particles are stationary and possess a uniform temperature. One active particle is given an elevated temperature of 60 °C. Because the fluid heats up due to transfer of heat from this active particle whereas all downstream particles can heat up as well the heat balance for all passive particles is solved as well to obtain the new \((n+1)\) particle temperature according to:

\[ T_{p,i}^{n+1} = T_{p,i}^n + \frac{Q_i \times \Delta t}{(\rho C_p) t} \]  

(14)

where \( Q_i \) is the computed heat flux of particle \( i \):

\[ Q_i = -\iint_{S_{p,i}} (k_f \nabla T_f - \bar{n}) \, dS \]  

(15)

Because in experiments all passive particles are copper beads \( \text{Bi}<1 \) and Eq. (14) is valid. Other boundary and initial conditions include a prescribed velocity and temperature at the inlet, zero gradient conditions at the outlet and either free-slip or periodic boundaries on the laterally confining walls. Other fluid properties and domain characteristics are given in Table 2.

3. Setup

The setup consists of two sections, see Fig. 4. The bottom section consists of a mass-flow controller with a capacity of 400 l/min and a distribution chamber, with a monolithic membrane to allow for uniform gas distribution. The top section consists of a square channel with holes drilled in two opposite sidewalls, to allow for the abacus-like structure of the arrays. The top section is interchangeable to allow for mounting different top sections and thus studying systems at different solids volume fractions. A thermocouple is attached to the top section to measure the ambient temperature \( T_{\text{amb}} \). A pressure sensor is mounted to the side (Dwyer MagneSense with a range of 0–1250 Pa).

The sizes of the top sections are given in Table 3. The height of the top section is 0.1 m. The 1 mm holes are organized in a matrix of 11 rows of 9 holes. Strings of 0.12 mm are strung through the holes, each row is offset by half a particle diameter, such that the beads can be strung in a body-centred cubic (BBC) arrangement. The particle arrangements have been strung by eye and as such can have a small deviation from an ideal BCC arrangement.
4. Results

In total three different systems are studied with both the CTA probe and the DNS simulations. The first system is the heat transfer of a single particle in unbounded flow, which is described by the Ranz–Marshall equation. The second test consists of an inline array of three particles, which allows the study of shielding and the effect of inter-particle distance on the fluid particle heat transfer. The third and final system consists of semi-structured arrays of particles with varying porosity, most closely resembling the conditions for which the Gunn correlation was established. The experiments are performed in the relevant range of Reynolds numbers, i.e. $0 < Re \leq 1500$. In the simulations the range of Reynolds numbers is limited $20 < Re \leq 400$, because of computational limitations. In the semi-structured arrays the range of Reynolds numbers is sometimes smaller due to either, avoidance of fluidization or limitations of the probe.

4.1. Single particle

Fluid-particle heat transfer for a single particle in unbounded flow is a fairly well-defined system and follows the well-known Ranz–Marshall equation. In the experiments the CTA-probe is put inside an empty top section. The inflow is varied so that the particle Reynolds number is varied between 0 and 1500 ($Re=1500$ is the maximum capacity of the probe). All experiments are averaged over time to eliminate periodic changes. For the simulations a particle is placed inside a box of 5 by 5 by 15 times the particle size. The particle is placed at $3 \times d_p$ from the inflow boundary. A free-slip boundary condition is used for the sides of the domain. A grid dependency study was done to check whether the simulations are grid independent. It was found that at 20 grid cells per particle diameter these simulations are nearly grid independent; deviation of 2%.

The results are shown in Fig. 5. It can be seen that the experiments are in very close agreement with the Ranz–Marshall equation. The deviation is about $\pm 5\%$, except for the lower Reynolds regime, where other heat transfer mechanisms play a role. In general the simulations follow a similar trend and possess deviations of $\pm 20\%$. Extrapolation of the results however would lead to extensive deviations with experiments and the Ranz–Marshall equation. Possibly the transient behaviour of the wake, as visible in Fig. 6, leads to an unsteady solution in the DNS results for higher Reynolds number flows.

Fig. 6 shows the temperature fields around the single particle for different Reynolds numbers. It can be seen that the wake is axi-symmetric and stable until $Re=200$ and shows slight instability from $Re=300$, when the toroidal vortex breaks and a vortex street is formed, Johnson and Patel (1999).

The deviations of the experiment all results from the Ranz-Marshall at small Reynolds numbers and can be explained by the existence of other important modes of heat transfer; free convection, radiation, conduction from probe to support and convection/conduction of exposed wire in between the probe and the support. These mechanisms have been schematically represented in Fig. 7.

Table 4 sums the approximate contributions of these mechanisms. Summation of all heat transfer mechanisms leads to a total of 109% of the found experimental value at $Re=0$.

It is also possible to express the combined effect of forced and free convection in terms of the Nusselt number following the work of Yamanaka et al. (1976):

\[
\text{Nusselt Number} = \frac{q L}{\alpha A (T_p - T_f)}
\]

where $q$ is the heat flux, $L$ is the characteristic length, $\alpha$ is the thermal diffusivity, $A$ is the area, and $T_p$ and $T_f$ are the particle and fluid temperatures, respectively.

Fig. 5. Nusselt number as a function of Reynolds number for a single particle in unbounded flow.

\[ \text{Nu} = 2 + \left( \frac{126 \text{Re} + 57 \text{Re}^{3/2} \text{Pr}^{1/3}}{1 + 52 \text{Re}^{3/2} + 100 \text{Re}} \right)^{2/3} + \left( 0.44 \text{Gr}^{1/4} \text{Pr}^{1/4} \right)^{2/3} \]

Fig. 8 shows the ratio of the obtained Nusselt numbers from combined convection (Eq. (16)) over the Ranz–Marshall equation and the experiments in Fig. 5. It can be clearly seen that forced convection makes up over 95% of the heat transfer when \( Re > 400 \). It can also be seen that our experiments match to within 2–3% with the work of Yamanaka et al. (1976) except at \( Re = 0 \), which can be attributed to radiation and conduction to the probe support respectively.

4.2. Inline array

The next system studied concerns an inline array of 3 particles. This system is used to study the effect of shielding and inter-particle distance on the fluid-particle heat transfer. The system resembles the system in the study of Tavassoli et al. (2013).
Fig. 9. Overview of the inline array setup.

Fig. 10. Left: Measured Nusselt number of the 2nd particle in an inline array of 3 spheres as a function of the Reynolds number for different inter-particle distances. Right: Measured Nusselt number normalized by the Nusselt number from the Ranz–Marshall experiment for different Reynolds numbers and inter-particle distances.

difference between these two systems is that here only one active particle is used (i.e. the particle on the CTA probe). The other two particles are initially cold but can heat up because of heated fluid passing the particle. In the simulations this is incorporated using Eq. (14). In simulations only an inter-particle distance of 2 is used. In the experiments the effect of inter-particle distance is varied to study both the effect of shielding and the inter-particle distance.

The setup is schematically represented in Fig. 9. The first particle is situated at $3 \times d_p$ from the inflow boundary. The active particle is placed in the second position and in experiments the particle–particle distance is varied between 2 and $6 \times d_p$. In the simulations both the side walls are placed at 2.5 $d_p$ from the centre of the particles and in the wake a domain size of $10 \times d_p$ is available. For $Re > 400$ the side walls are placed at $3.5 \times d_p$, because otherwise the transient behaviour of the wake there would introduce confinement effects. The simulations have been performed with 20 grid cells per diameter, except the simulation at $Re=600$, which is performed with 30 grid cells per diameter to resolve the thinner boundary layers around the particles.

Fig. 10 shows the experimental results for the inline array of 3 spheres with varying inter-particle distances. The results clearly show an increased Nusselt number over the full Reynolds range up to $Re=1400$. This enhancement effect is largest when the spheres are close to each other and diminishes with increasing inter-sphere distance. From an inter-particle distance larger than 6 no enhancement was found, just as reported by Reddy et al. (2013). The right figure shows the relative increase in Nusselt number from the inline array with respect to the results of a single particle in unbounded flow (the Ranz–Marshall experiment). From Fig. 10 it can be seen that the obtained Nusselt numbers follow a different trend for $Re < 400$ than for cases with $Re > 400$. This might be explained due to the contributions of other heat-transfer mechanisms, mainly free convection, which still has a substantial effect up to $Re=400$, see also Fig. 8.

Fig. 11 shows the results of an inline array of three spheres with an inter-particle distance of $2 \times d_p$. The experiments show an enhancement of the Nusselt number with respect to the Ranz-Marshall equation. The simulations however clearly show a shielding effect for the lower Reynolds regime, $Re < 400$ and an enhancement for $Re > 400$. This shielding effect was expected because of the sphere in front of the active sphere, which was also reported by Tavassoli et al. (2013). The enhancement seems to appear at Reynolds numbers exceeding 400. From simulations it can be seen that the enhancement is accompanied with an onset of a transient wake following the first sphere. This transition is also shown in Fig. 12. Up to $Re=300$ the space between the first and second spheres is slowly heating. From $Re=400$ however the fluid between the two spheres is constantly refreshed because of the transient wake behaviour. In experiments the onset of a transient wake might appear earlier, either by effects of free convection or possibly also because of the inherent flow instabilities from the mass-flow controller or the distributor.

4.3. Semi-structured arrays

In the last case discussed in this paper, experiments have been performed to study the effects of solids volume fraction and Reynolds number on the fluid-particle heat transfer. The work of Gunn, Eq. (3), is used as a reference. The top sections as described in Table 2 are used. The solids volume fraction ranges from 0 to 0.6 with increments of 0.1 and Reynolds numbers ranging from 0 to 800, with two exemptions. At a solids volume fraction of 0.6 (packed bed condition) the Reynolds number is limited to 200 because of fluidization of the particles at higher flow rates, which
would damage the probe. At a solids volume fraction of 0.1 the Reynolds number is limited to 500, because of limitations of the mass flow controller. Five rows in the top section are strung with beads, except for the third row where space is reserved to position the probe.

The simulations have been performed with very similar structures. All particles are placed in a BCC packing, except for a solids volume fraction of 0.6 which was simulated as a random packing, similar to what was done in the experiments. All simulations are first performed at a grid resolution of 20 to initialize the flow field. After some time the simulation data is interpolated to provide initial conditions for simulations at a finer grid of 40 grids per diameter. Only five rows and five planes of particles are used, giving a total of 125 particles. The simulations have in- and outflow conditions as explained before. In this case however periodic boundary conditions were applied in the lateral directions. Only the centre particle has an elevated temperature of 60 °C, all other particles are allowed to heat up by the surrounding fluid.

Fig. 13. Nusselt number as a function of the particle Reynolds number for several different porosities and a parity plot of the results from experiments and simulations.
comparatively large deviation due to combined heat transfer effects as a 20% error margin. At a lower Reynolds number there is also a higher packing fraction. The parity plot of the simulations and probe support starts to interfere in the local solids distribution at an underestimation of the heat transfer. For the experiments the fraction of 0.5. It might be that for the simulations the grid resolution becomes a problem at higher solids loadings, resulting in an underestimation of the heat transfer. For the experiments the probe support starts to interfere in the local solids distribution at higher packing fractions. The parity plot of the simulations and experiments shows however that most of the results fall within a 20% error margin. At a lower Reynolds number there is also a relatively large deviation due to combined heat transfer effects as discussed in the section on a single particle in unbounded flow.

**Fig. 14** shows a parity plot of the Nusselt numbers obtained with experiments in comparison to the Gunn correlation. Again it can be seen that most experiments fall well within 20% error margins. This shows that the use of CTA-probes in combination with a semi-structured setup is very well suited in studying heat transfer of different packings, thus covering the full range of the Gunn-correlation. The simulations using a IB/DNS method are in very close agreement with the experiments and hence also with the Gunn correlation. This is unlike the work of Tavassoli et al. (2013), where a structural underestimation of the Gunn correlation was found. There are of course very distinct differences in the system under study. In the work of Tavassoli et al. (2013) a fully active and random packing is used, which necessitates the use of averaging the temperature in the driving force and over all particles. On the other hand, in this work a single active particle in structured arrays is used, with a very clear driving force and local structure. This might well be related to the differences found in use of the different forms of the driving force in calculating the Nusselt number as studied in the book of Bird et al. (2007).

**5. Conclusion**

A Constant Temperature Anemometer (CTA) was successfully reconfigured to act as a heat-transfer probe in gas-particle flow systems. In combination with an abacus like semi-structured bed the effects of Reynolds number and solids packing fraction on the Nusselt number was successfully studied. First CTA measurements were performed on a single particle in unbounded flow and compared to the well-known Ranz–Marshall equation as well as Direct Numerical Simulations using an Immersed Boundary method. Second, the CTA technique was used to study the effects of particle shielding and inter-particle distance, showing that the technique is best equipped to study inline arrays of spheres beyond \( Re=400 \) where the onset of a transient wake drives the enhancement of the Nusselt number with respect to a single sphere. Third, experimental and numerical results for different solid fractions show a remarkable good comparison to the Gunn correlation, this unlike results from fully active arrays of particles in DNS, that show a distinct underestimation with respect to Gunn. Further study into the proper use of the heat transfer closure law might be studied using a DNS of a small fluidized bed as in the work of Deen et al. (2012). Also the use of coupled particle image velocimetry and infrared imaging as by Patil et al. (2015) might give more insight into the use of closure laws in fluidized bed systems.

**Acknowledgements**

This research was funded by the European Research Council, under the Advanced Investigator Grant Scheme, Contract no. 247298 (Multiscale flows). This paper is an extension of a conference paper that was presented at the 11th International Conference on CFD in the Minerals and Process Industries (CFD2015), and was nominated for submission to Chemical Engineering Science based on its designation as a high-quality paper of relevance to chemical engineering.

**References**