



Experimental investigation of particle flow in a gravity -driven moving bed electric heater: improved methodology and preliminary geometry comparison

Taras Koturbash, Ata Ülger, Parth Kumavat, Salvatore Guccione

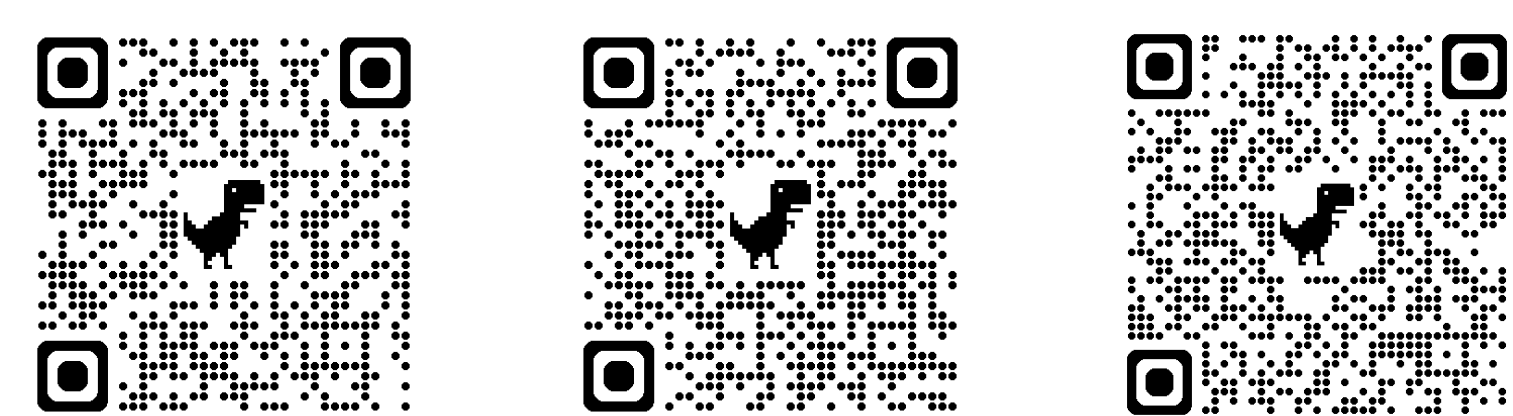
Particle-based Concentrating Solar Power (CSP) systems enable high operating temperatures and thermal energy storage using inexpensive solid materials. Within the POWDER2POWERproject, an electric moving-bed heater is being developed to further increase the temperature of olivine particles using renewable electricity. Gravity-driven shell-tube heat exchangers stand out among other solid-particle HX designs due to their low cost, reduced parasitic losses, and ease of integration into various processes. A potential method for achieving higher particle loop temperatures involves incorporating a superheater after the solar receiver, a concept explored in the MW-scale CSP prototype within POWDER2POWERproject.

Background

In a gravity-driven heater, particles flow around horizontal heating elements. Conventional cylindrical elements generate characteristic stagnant zones above the tubes and void zones below them, reducing particle-heater contact and potentially creating local overheating.

The objective of this work was to further develop the experimental methodology and investigate whether an airfoil-inspired heater geometry can improve particle flow around the heating elements.

Live recordings of experimental runs



Experimental test rig

A transparent cold-flow test rig was developed to reproduce the particle flow through a section of the moving-bed electric heater.

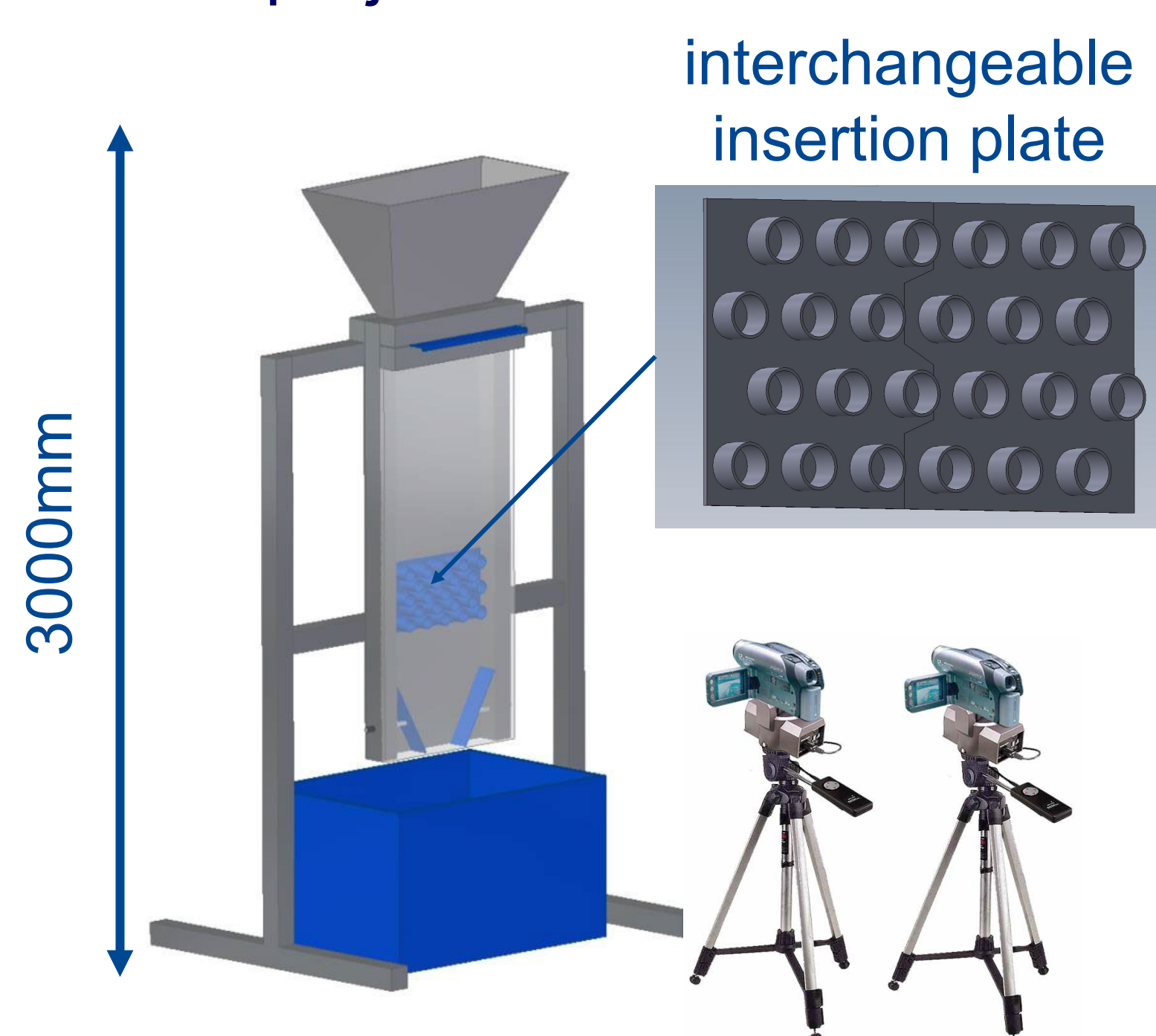
The rig consists of:

- transparent PMMA test section;
- interchangeable heater-element inserts;
- approximately 150–200 kg olivine inventory;
- gravity-driven continuous particle flow;
- redesigned adjustable outlet for improved control of the particle discharge rate;

Improved optical measurement procedure

A two-camera arrangement was investigated to obtain both a wider view of the particle bed and a more detailed view of the flow around individual heater elements.

Different lenses (16MM, 25MM, 35MM, 50MM) and camera positions were tested to improve: field of view, spatial resolution, and particle visibility for PIV analysis without any addition of tracing agents/particles.



Investigated heater geometries

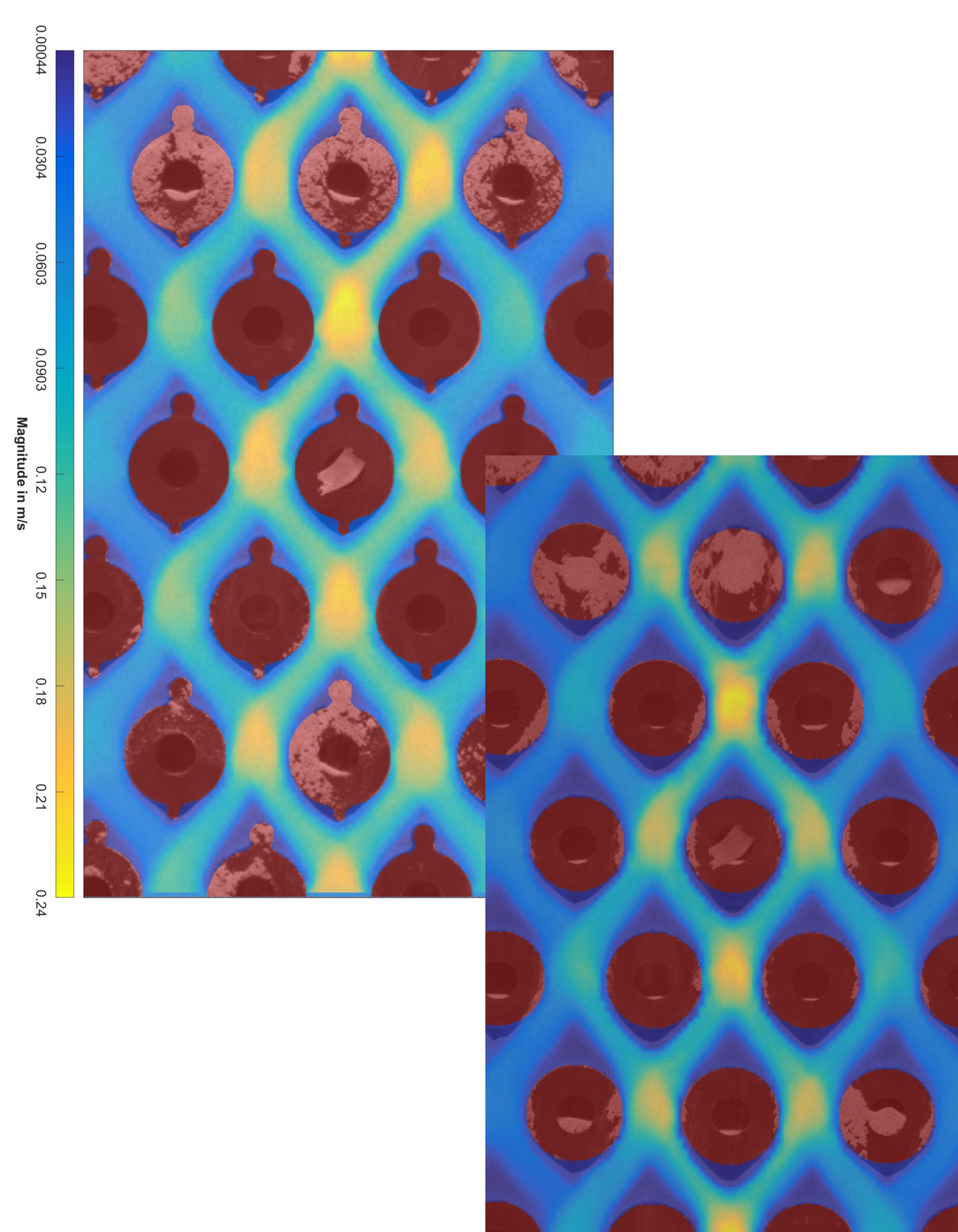
- Reference cylindrical element

Conventional circular OD=51mm heater geometry representative of the tubular electrical heating elements considered for the POWDER2POWER heater.

- Airfoil-inspired element

A modified cylindrical geometry consisting of the reference cylinder combined with an additional rod of diameter $D/4$ attached to the upper/upstream and $D/8$ do the bottom/downstream side of the cylinder.

The purpose of this simple modification was to change the way particles approach and move around the heating element without requiring a completely new heater design.



Want to know more?



Preliminary experimental results

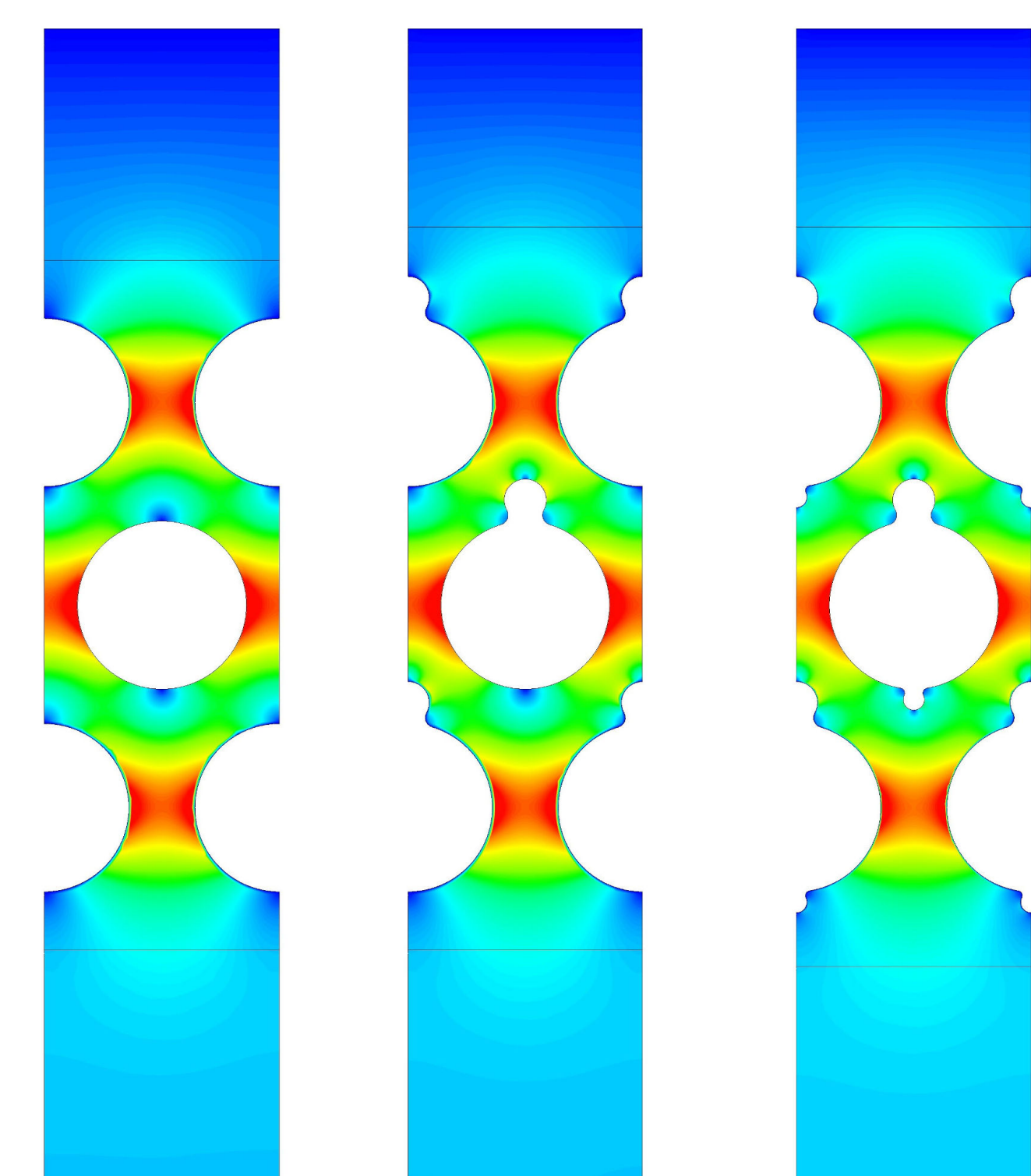
Cylindrical geometry:

- formation of a stagnant region upstream/above the element;
- formation of a particle-depleted void region downstream/below the element;
- significant acceleration of particles around the sides of the cylinder;

Airfoil-inspired geometry:

- smoother redirection of particles around the element;
- reduced abrupt particle acceleration around the sides;
- improved particle-flow uniformity close to the heater surface;
- reduced tendency toward pronounced stagnant and void regions.

The results therefore indicate that relatively simple modifications of the heater cross-section can significantly influence granular flow behavior.



Preliminary CFD investigation

A numerical model of the particle flow was also investigated in ANSYS Fluent as a complementary approach to the experiments.

The work demonstrated several challenges associated with representing an effectively continuous moving bed, particularly: maintaining consistent inlet/outlet particle mass flow, defining appropriate boundary conditions and reproducing the experimentally observed near-wall particle behaviour.

At the present stage, the numerical solution is therefore considered exploratory only and is not used for quantitative validation of the experimental results.

Further model development and experimental validation are required before CFD can be applied reliably for geometry optimization.