Proceedings of 4th International Topical Meeting on High Temperature Reactor Technology
HTR 2008
September 28-October 1, 2008, Washington, DC, USA

HTR2008-58143

INTEGRATED SYSTEMS CFD MODELLING APPLIED TO DIFFUSION-BONDED COMPACT HEAT EXCHANGERS

Jan-Hendrik Kruger*
Louis A. le Grange

Postgraduate School for Nuclear Science and Engineering
North-West University, Private Bag X6001, Potchefstroom, 2531, South Africa
Email: jhk@mtechindustrial.com

Gideon P. Greyvenstein
PBMR (Pty.) Ltd., 1279 Mike Crawford Ave, Centurion, 0046, South Africa

ABSTRACT

Micro-channel heat exchangers consist of a number of plates, each containing fluid channels etched in the surface and diffusion bonded together to create a porous core of metal. The primary and secondary sides of the exchanger are formed by connecting the channels on alternating plates to the respective leader pipes. To analyze the thermal response of exchangers during operation, simulation software is used to create a network of numerical models representing the real-life thermal-hydraulics components. The Systems CFD approach uses one-dimensional empirical models for the fluid flow inside the channels and a three-dimensional model for the heat distribution inside the core. Spatial analysis of the geometry gives a connectivity stencil between the one- and three-dimensional models. This stencil implicitly links the equations of the models at matrix level in the numerical solver, with faster convergence in fewer iterations than when the models are coupled explicitly in different software applications. Results presented show the heat flux through an exchanger core and the fluid flow inside the channels.

INTRODUCTION

Diffusion-bonded heat exchangers represent a new generation of compact exchangers for use in the nuclear and other process industries. Its construction consists of a large number of thin (between 1 and 2 mm thick) metal plate layers containing micro channels and welded together by diffusion bonding to form the exchanger. These exchangers have a small physical footprint, are thermally very efficient and are especially suitable for high temperature applications. The use of diffusion bonding as welding process further allows exotic materials (e.g. titanium and nickel) to be used in heat exchangers that would otherwise be difficult to manufacture using standard welding techniques and shell-and-tube designs. Exotic materials are especially desirable in situations where the fluids in the exchanger might be corrosive and chemically react with the metal of the exchanger.

The modern nuclear power plant is an example where compact heat exchangers can be used with great success. The designers of the Pebble Bed Modular Reactor (a Generation IV gas-cooled reactor currently under development in South Africa) use the Flownex Systems CFD (Computational Fluid Dynamics) code to simulate the transient behaviour of the thermal-fluid networks in the reactor and power plant. The Flownex software uses semi-empirical one-dimensional models for thermal fluid components like the reactor, piping network, pumps, heat exchangers and control valves [1]. While such 1D-based models are fast, memory efficient and accurate, fatigue and response analysis of the exchanger requires more accurate 3D-based calculations of the heat flux inside the recuperator and the exist-

*Address all correspondence to this author.

ing 1D-element based models of exchangers are not sufficiently detailed. To better understand the transient effects of thermal shocks and material fatigue on the new designs, innovative new models are needed to capture the operational behaviour of exchangers while still being part of the complete thermal-hydraulic system and maintaining speedy solution times.

The Systems CFD approach is a general methodology that integrates a combination of 1D and 3D modelling techniques to create a hybrid model of a complex component. The approach has already been proven as a successful modelling strategy through its use in existing Flownex components, a relevant example being the current pebble bed reactor model [2].

The application of Systems CFD to micro-channel heat exchangers takes the method a step further by using the actual physical geometry of the exchanger for the 3D side and not an idealized, simplified version. The 1D elements are used to model fluid flow inside the micro channels while detailed 3D CFD techniques for porous media are used to calculate the heat flux inside the metal layers of the exchanger. The micro channels are traced through the CFD cells to calculate an effective porosity and to determine the connectivity between 1D elements and 3D cell volumes. Through the connectivity stencil the coefficients of the 1D and 3D energy equations are implicitly linked at matrix level in the solver. The advantage of directly coupling the matrices is faster convergence with fewer iterations than when the information is exchanged externally between models running in different software applications.

In the following sections, the physical construction of a micro-channel heat exchanger is described and how the Systems CFD methodology is applied to create the integrated model of the exchanger. The last section showcases some typical results that illustrate what can be achieved through the use of this method.

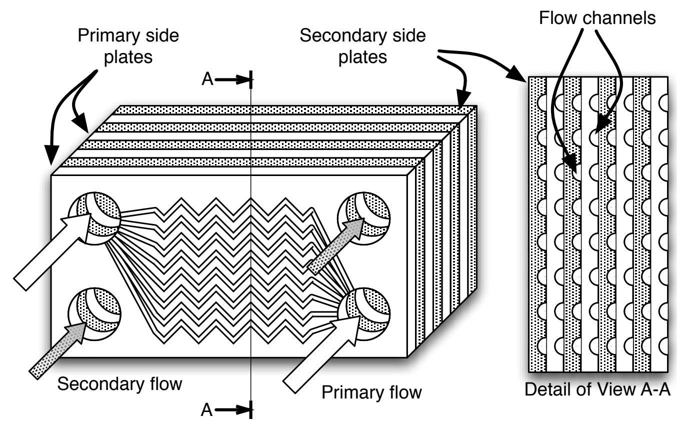

Figure 1. EXAMPLE OF COMPACT HEAT EXCHANGER CORE SHOWING LAYERED PLATES AND MICRO-CHANNELS

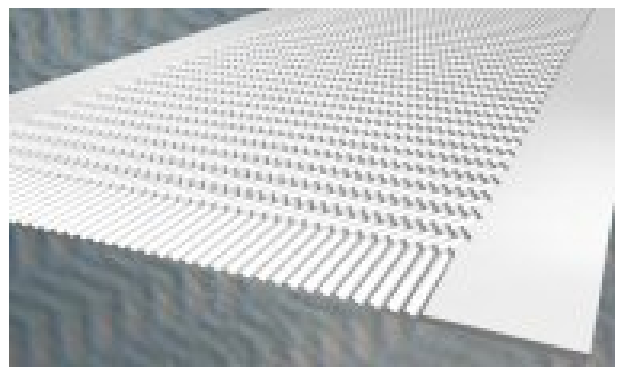

Figure 2. PLATE WITH CHANNELS ETCHED IN THE SURFACE [3].

PHYSICAL CONSTRUCTION

The layout of a typical micro-channel heat exchanger (MCHX) is shown in Fig. 1. Cross- and counter-flow pathways are commonly used as basis for the design of fluid flow through the primary and secondary sides. Depending on the size and design requirements for the equipment, channels can be etched (Fig. 2), laser cut or machined into the plates. In other cases, the plates can also be machine pressed to form the channels (Fig. 3).

The individual plates are stacked upon each other by alternating the primary and secondary side plates as shown in Fig. 4 and then welded together with diffusion bonding to obtain a porous chunk of material containing the channels. Diffusion bonding eliminates the separation between subsequent plate layers in such a manner that there is even grain growth (Fig. 5) across the interface and thus no contact resistance. This results in very efficient heat transfer between the channels and across the different primary and secondary side layers because there are no air gaps between plates. The resulting form factor of a diffusion-bonded exchanger is also markedly smaller for the same thermal loading when compared to the physical size of a traditional shell-and-tube or plate exchanger.

SYSTEMS CFD METHODOLOGY

To simulate a heat exchanger using the Integrated Systems CFD methodology, the physical model must first be discretized with the relevant 1D or 3D approximation. The resulting numerical grids (the detailed 3D mesh of the solid material and the 1D pipe network representing the channels) determine the coefficients that are linked together implicitly. This integrated system of equations is then solved by the matrix solver as a single system of unknowns.

General algorithm

The general method for establishing the integrated model of the heat exchanger is described in this section. The steps include

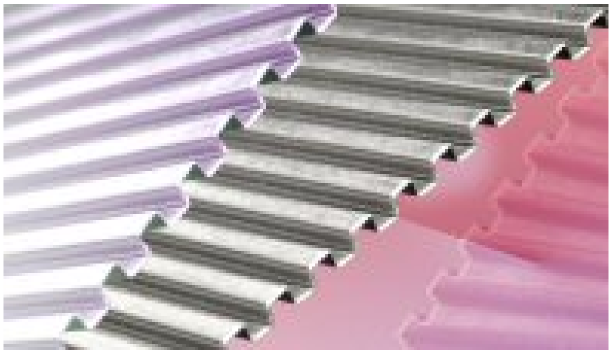

Figure 3. CORRUGATED PLATES WHERE THE PLATES ARE MACHINE PRESSED TO FORM THE FLOW CHANNELS [3].

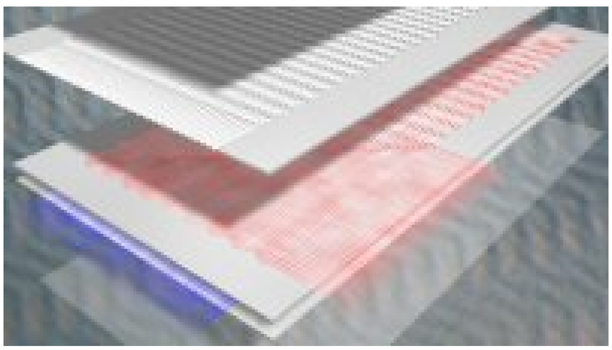

Figure 4. DIFFERENT PLATE LAYERS ARE STACKED UPON EACH OTHER TO FORM PRIMARY AND SECONDARY SIDES [3].

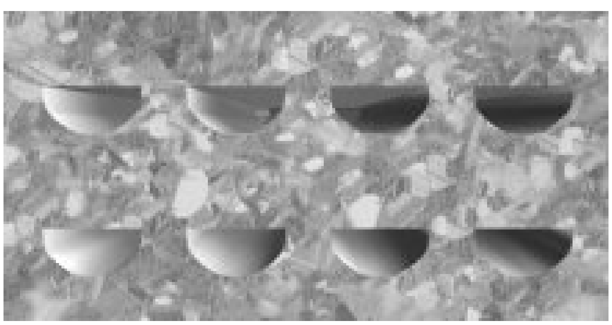

Figure 5. DIFFUSION BONDING PROMOTES GRAIN GROWTH BETWEEN PLATES FOR IMPROVED HEAT EXCHANGE [3].

the design of the physical exchanger, meshing the solid portion, tracing the channels, creating the pipe network, linking the pipe network and solid mesh and solving all the fluid flow and energy equations as coupled system. In the sections following this summary, the different steps are described in more detail.

Model design and generation: The exchanger core and channels are designed by using the custom geometry design utility or any other computer aided design (CAD) software package to obtain a full 3D solid geometry representation of the heat exchanger.

Mesh the core: The solid bulk of the model is extracted from the geometric model. The meshing algorithm uses unstructured tetrahedral cells ("tets") to obtain an initial mesh for the domain, from where the tets are then combined to create polygonal cells for a more efficient numerical grid.

Trace the channels: The entities describing the channels are extracted from the original geometric model. These micro-channel objects are overlaid on the 3D CFD mesh and traced from beginning to end to determine all the intersection points between the channel objects and cell faces in the numerical mesh. The base object is split into separate sections for every cell the channel passes through and these sections are linked to their parent cell. This tracing step creates the sections that are later used in the creation of the pipe element network and establishes the connectivity matrix between 1D and 3D elements.

Create pipe elements: The tracing step creates a channel section for every cell the micro-channel intersects. For every channel section contained in a cell, a pipe element with the same length and inlet/outlet diameters as that section is automatically generated and linked to the rest of the pipe element network.

Link pipes and cells: 1D (pipe elements) and 3D elements (solid cells) are linked together with dedicated heat transfer elements. The cell temperatures are linked to the fluid temperatures inside the pipes through automatically updated convection coefficients. The convection coefficients are continuously updated based on the type of fluid inside the pipes and the flow conditions. The links are based on the connectivity matrix calculated during the channel tracing step.

Solve integrated system: All coefficients are assembled in the same matrix system to solve the fluid-flow and energy equations for all of the 1D and 3D elements as an integrated system. Using a linked solution algorithm gives immediate numerical feedback between 1D and 3D elements. This results in faster convergence using fewer iterations than a method that explicitly exchanges information between separate 1D and 3D models each with their own equation matrix and solver.

Thermal conduction network

The exchanger is designed using custom software that automatically generates the geometry of the core and micro channels according to certain pre-set design parameters. (An example of this process is the model generated for the demonstration case, shown in Fig. 6). The core of the exchanger is then extracted

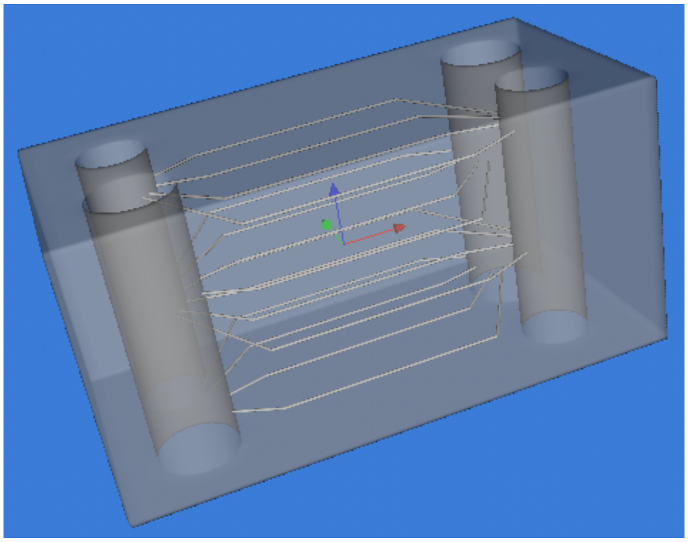

Figure 6. DEMONSTRATION HEAT EXCHANGER GEOMETRY

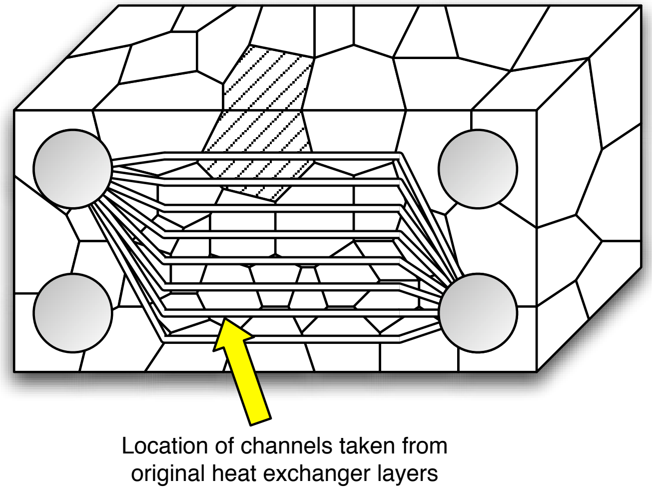

Figure 7. CFD MESH OF SIMPLIFIED HEAT EXCHANGER WITH TYPICAL CELL SHOWN IN CROSS HATCHING

as a solid geometry model, taking into account the primary and secondary side inlets and outlets (leader pipes) running down the length of the exchanger, but discarding the detail of the channels. The effects of the fluid volume contained in the micro channels on the heat flow through the solid, are approximated by defining porous conductive properties for the bulk material.

The core of the heat exchanger (excluding the volumes occupied by the primary and secondary leader pipes) is meshed using a typical unstructured polygonal algorithm. In Fig. 7, an illustrative representation of the CFD grid is shown. By including the bulk material, the thermal inertia of the exchanger is included in the component model. This in turn improves the prediction of the overall thermal response characteristics of the exchanger when included as part of a larger system simulation.

To calculate the heat flux distribution inside the core, the standard energy equation is modified to use porous conductivity and discretized on the unstructured mesh with a face-based approach, see [4]. The conductive heat transfer between 3D cells can be represented by a network of electrical resistances, as depicted in Fig. 8. By adopting this representation, the integration between the pipe element network and the conduction network can be conceptualized in a more natural manner. The values of these "resistances" are nothing else than the coefficients obtained from the CFD mesh and contain terms describing the distances between cell centres, the connecting face areas, the material's thermal conductivity (determined on a per-cell basis due to varying porosity) and the amount of non-orthogonality between neighbouring cells.

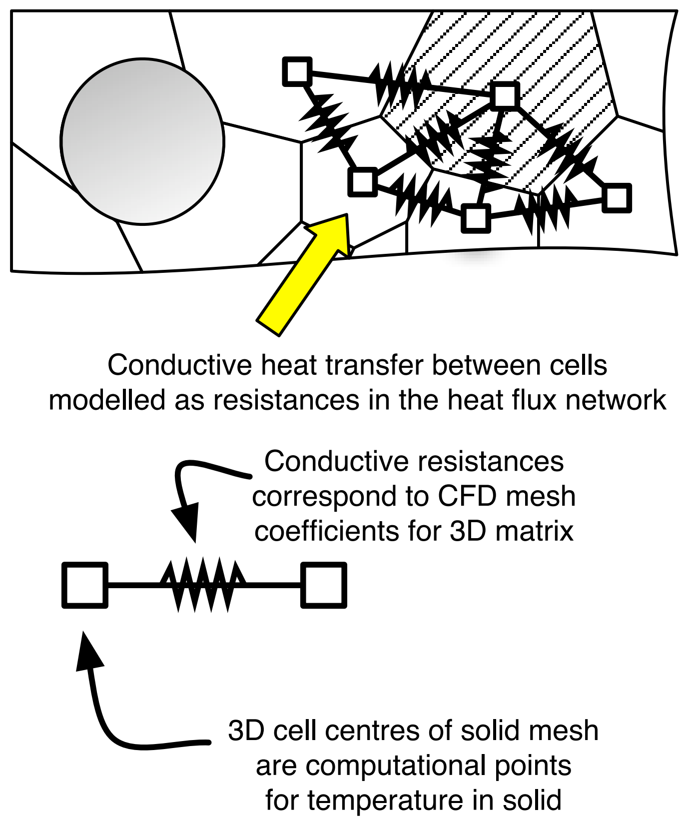

Figure 8. HEAT FLUX BETWEEN 3D CELLS IN EXCHANGER CORE

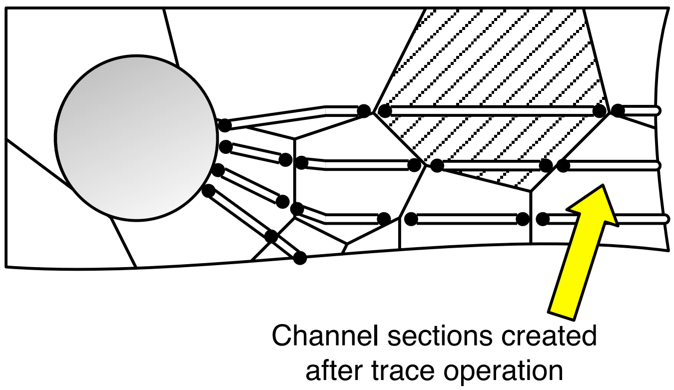

Figure 9. DETAIL OF CFD MESH WITH TRACED CHANNELS SUPERIMPOSED ON THE GRID

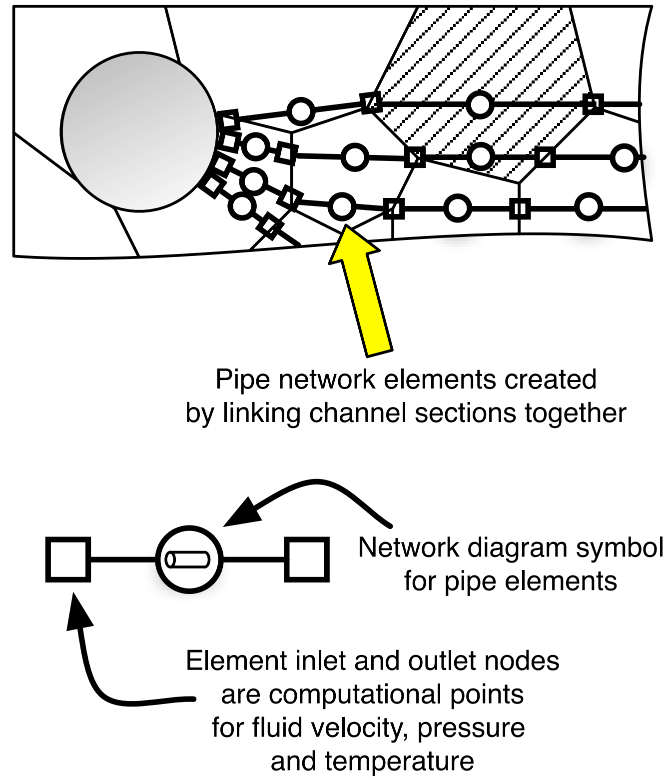

Figure 10. PIPE ELEMENTS CREATED FOR CHANNEL NETWORK

Pipe element network for micro channels

The micro channels are individually traced through the CFD cells to determine the connectivity matrix between cells and channel sections. (The results of a typical trace operation are shown in Fig. 9). While tracing the channels, the exchanger geometry (as originally designed) is used to obtain the flow areas and lengths/volumes of every section of channel that passes through a cell. These sectional properties are used to define the porosity for each cell (the ratio of channel void volume to cell volume) and to modify the thermal conductivity for the cell.

In Fig. 9, every micro channel is represented by a series of separate channel sections, but connected to each other at cell faces. The channel sections from the trace operation are used to create a flow network containing the corresponding number of one-dimensional pipe elements. By using the same connectivity matrix as the channel sections, the pipe elements are then linked to the cells in the CFD mesh. At the endpoints of the flow channels, the inlets and outlets are connected to the fluid flow in the primary and secondary feeder pipes. The creation of pipe flow elements is shown in Fig. 10 where it can be seen that the endpoints of elements coincide with the location of channel and cell face intersections.

Combined discretization model

The typical structure of the various links between the network model and the CFD mesh are shown in Fig. 11. The primary and secondary layers link to each other through the temperature node of the CFD cell. This cell centre node is used to link all the pipe segments contained in that specific cell's volume. It is tacitly assumed that the region defined by the cell is approximated by a single average temperature, which is consistent with one of the fundamental features of the finite-volume method [5].

When the flow channels are mapped on the CFD cells, the channels are subdivided into lengths with a certain inlet and outlet area (and different hydraulic diameters). This information is then used to complete the geometric properties of the pipe network for the exchanger. Numerical stability in the fluid momentum equations of the network model can be improved by combining very short pieces of pipe (e.g. when a channel barely clips a cell) into a longer pipe without affecting the numerical accuracy of the simulation.

The pipe elements link to the cell elements using specialized heat transfer elements with convection coefficients that are calculated dynamically depending on fluid flow conditions inside the pipes. The CFD cells are linked to each other through the conductivity coefficients and the energy equation for porous solid media is then used to solve for the temperature distribution inside the solid.

When all elements have been linked, the nodes indicated by the square markers in Fig. 11 form the unknown variables in the matrix system. The coefficients from the combined 1D and 3D models are combined in a single matrix to obtain a coupled solution for the unknown nodal temperatures. The solution algorithm of the network simulation software uses a segregated approach, where the fluid momentum, pressure correction and energy (temperature) equations are solved in sequence and the process is iterated to obtain a convergent solution [1].

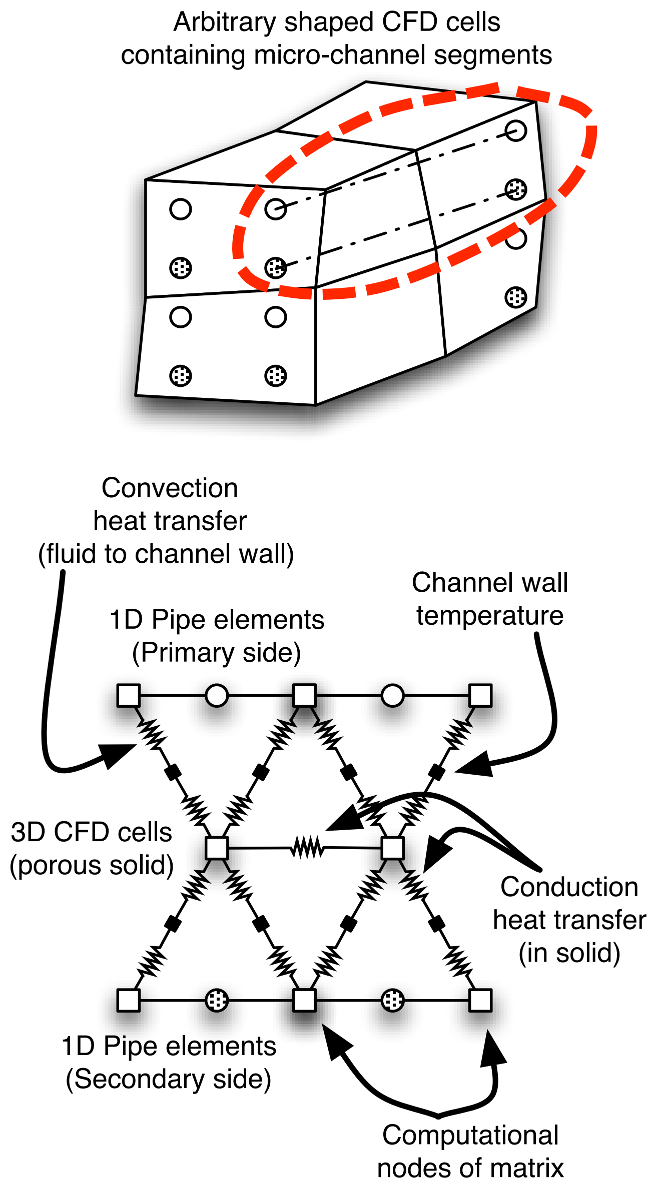

Figure 11. CONNECTIVITY BETWEEN THE 1D PIPE ELEMENTS AND THE 3D CFD CELLS IN THE HEAT EXCHANGER CORE

DEMONSTRATION CASE

In Fig. 6 the heat exchanger geometry used in the study is shown. Because the focus of this paper is not on the presentation of detailed results but rather on the description of the Systems CFD approach as modelling technique, a very simplified design was chosen as demonstration case. The design contains all of the key elements found in a typical compact exchanger, but greatly reduces the level of complexity in order to describe the technique and possible results with greater clarity.

Table 1. PHYSICAL DIMENSIONS OF HEAT EXCHANGER

Description	Value	Units
Width	400	mm
Height	200	mm
Length	200	mm
Primary leader	30	mm
Secondary leader	25	mm
Channel diameter	2	mm
Channels per layer	4	[-]
Number of layers	4	[-]

Case Description

The exchanger core consists of a rectangular block of metal with primary and secondary leader pipes running completely through the solid. There are four layers of circular channels (representing four stacked plates), with two layers as the primary (high pressure) side and the other two layers as the secondary (low pressure) side. The flow in the channels is configured in a counter-flow arrangement and the channels are directly connected to the leader pipes . Because diffusion-bonding is used between layers, the contact resistance between individual plates is negligible and the core can be modelled as a continuous porous solid containing the flow channels. The geometrical dimensions of the exchanger are described in Table 1 and the fluid properties of the primary side in Table 2 and that of the secondary side in Table 3. The boundary conditions and fluid properties are typical of the conditions in heat exchangers that are used in a helium-based direct loop reactor cycle.

Results

The core region of the exchanger was meshed with polyhedral cells, as shown in Fig. 12. As part of the meshing step, the size of the poly-cells are automatically adjusted so that there are more cells located in the vicinity of boundaries with high curvature (like the boundaries of the leader pipes). In Fig. 13 the pipe elements are shown that are created after completing the channel tracing step of the algorithm. These 3D objects are used for visual confirmation that the pipe network was created correctly and corresponds to the channel network exported to the network code for calculation purposes.

For the demonstration case, results are shown for the temperature distribution in the exchanger core (Fig. 14) and the fluid temperatures inside the flow channels, see Fig. 15. An inspection of the temperature distribution in the core indicates that the

Table 2. PRIMARY SIDE BOUNDARY CONDITIONS

Description	Value	Units
Fluid	High Pressure Helium	
Inlet temperature	423	K
Mass flow (per channel)	$2.0 \cdot 10^{-5}$	kg/s
Inlet pressure	90	bar
Density	6.13	kg/m^3
Viscosity	$3.59 \cdot 10^{-5}$	Pa.s
Thermal conductivity	$3.02 \cdot 10^{-1}$	W/mK
Specific heat	5195	J/kgK

Table 3. SECONDARY SIDE BOUNDARY CONDITIONS

Description	Value	Units
Fluid	Low Pressure Helium	
Inlet temperature	873	K
Mass flow (per channel)	$2.0 \cdot 10^{-5}$	kg/s
Inlet pressure	30	bar
Density	1.26	kg/m^3
Viscosity	$5.09 \cdot 10^{-5}$	Pa.s
Thermal conductivity	$4.09 \cdot 10^{-1}$	W/mK
Specific heat	5195	J/kgK

highest temperature gradients are located where the primary and secondary leader pipes are situated closer together. In the central region of the core where the active heat transfer area of the micro channels are located, the temperature contours spread out more gradually and this is also reflected by the temperature distribution inside the flow channels.

By using the Systems CFD technique, phenomena like a sudden change in inlet temperature can be modelled to quantify its effects on the core material and on the fluid conditions inside the channels. The transient behaviour of temperature waves in the core can be used to determine regions with higher material stresses and to gain more insight into how the system reacts to the thermal inertia of the exchanger. Together with that, the ability to calculate the fluid conditions inside the channels (mass flow, temperature, density, phase quality and convection coefficients) using emperical correlations, allows a much more accurate estimation of heat transfer between the metal and the fluid.

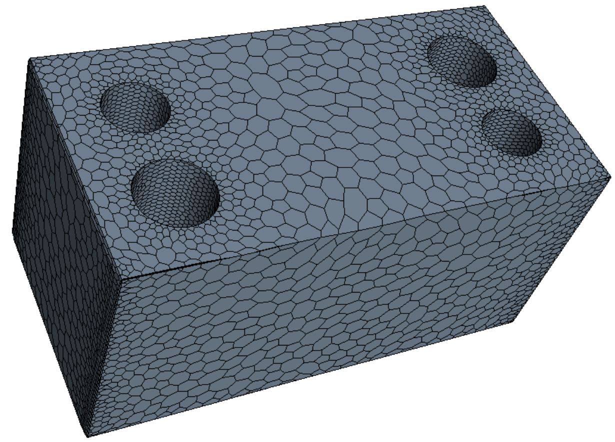

Figure 12. POLYHEDRAL MESH OF CORE REGION

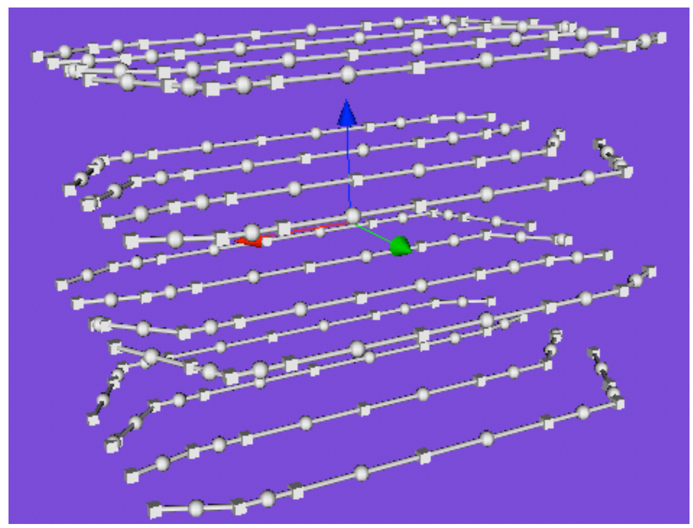

Figure 13. PIPE NETWORK OF CHANNELS

CONCLUSION AND FURTHER WORK

The application of the Integrated Systems CFD methodology to the modeling of compact micro-channel heat exchangers was described in this paper. The technique shows great promise in creating numerical models of thermal network components that enable more detailed analysis than one-dimensional models but at improved simulation speeds when compared with the use of detailed three-dimensional CFD models.

For the demonstration case, a simplified version of a compact heat exchanger with four header pipes and four channel lay-

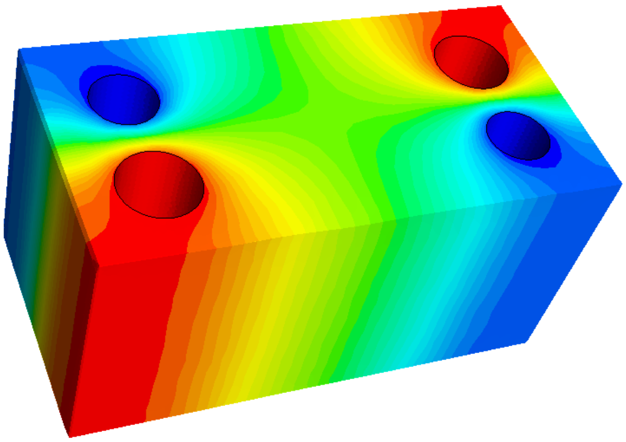

Figure 14. TEMPERATURE DISTRIBUTION INSIDE CORE REGION

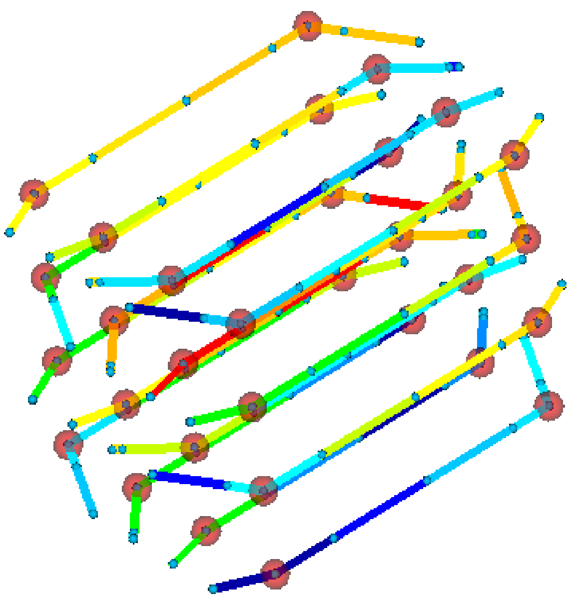

Figure 15. TEMPERATURES INSIDE FLOW CHANNELS

ers was described. The Systems CFD methodology allows the modelling of the heat flux distribution in the solid core of the exchanger while simultaneously coupled to the flow of the working fluid inside the channels. Future work will focus on sensitivity studies with regards to the 3D mesh resolution and its effects on convergence rate and solution accuracy. The effect of porosity on solid conduction calculations must be quantified through extensive verification with analytical and numerical models. Experimental validation against work done on the Helium Test Facility recuperator at Pelindaba in South Africa is planned as well as investigations into the application of the Systems CFD technique to other fields of study, e.g. the modelling of steam generators or the internal cooling channels inside turbine blades.

REFERENCES

[1] M-TECH INDUSTRIAL, 2006. *Flownex 7.0 User Manual*. See also URL http://www.flownex.com.

[2] van Antwerpen, H., 2007. "Modelling a pebble bed high temperature gas-cooled reactor using a System-CFD approach". PhD thesis, North-West University.

[3] Heatric, 2008. Diffusion-bonded heat exchangers. See also URL http://www.heatric.com, June.

[4] Ferziger, J., and Peric, M., 2001. *Computational Methods for Fluid Dynamics*, 3 ed. Springer.

[5] Versteeg, H., and Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics*, 2 ed. Prentice Hall.