Numerical modelling of the flow and heat transfer in a prismatic block VHTR single-channel fuel module

GJ Nel

orcid.org/0000-0002-1050-2659

Dissertation submitted in partial fulfilment of the requirements for the degree *Master of Engineering in Nuclear Engineering* at the North-West University

Supervisor: Prof CG du Toit

Graduation ceremony: May 2019

Student number: 24118699

ACKNOWLEDGEMENTS

Sincere thanks to the following persons for their support during this study:

- Prof. Jat du Toit, Study Leader
- South African Research Chairs Initiative of the Department of Science and Technology and National Research Foundation of South Africa, Funding
- Ockert Koekemoer, Simulation assistance
- Dr. Jean du Toit, Dissertation structure

PREFACE

- This is to state that I, Gert Johannes Nel, have chosen the article format for submitting my dissertation.
- The dissertation is written in South African English.
- Articles are written by:

GJ Nel, CG du Toit

South African Research Chairs Initiative of the Department of Science and Technology and National Research Foundation of South Africa (Grant No 61059).

Area, North-West University, Potchefstroom 2520, South Africa

Email: Gertt.nel@gmail.com, jat.dutoit@nwu.ac.za, ockie.koekemoer@gmail.com

- I, CG du Toit, hereby give permission that Gert J Nel may submit the article(s)/manuscript(s) for degree purposes.
- The peer-reviewed article (Ch 2): *A System CFD Model of a Single-Channel Fuel Module in a Prismatic Block VHTR* was published in the Proceedings of the 11th South African Conference on Computational and Applied Mechanics (SACAM2018).
- Acknowledgements for all articles:

We thank the South African Research Chairs Initiative of the Department of Science and Technology and National Research Foundation of South Africa and the North-West University for their financial support.

NOMENCLATURE

A	Area
Ai	Aicher number
C_p	Specific heat
D	Diameter
k	Thermal conductivity
Nu	Nusselt number
Nu_{FD}	Nusselt number for fully developed flow
P	Pressure
PD	Power density
PD_{max}	Maximum power density
Pr_b	Prandtl number based on bulk fluid temperature
Ra	Rayleigh number
Re	Reynolds number
Re_b	Reynolds number based on bulk fluid temperature
R_g	Gas constant
r	Radial position
T	Temperature
T_b	Bulk fluid temperature
T_w	Wall temperature
z	Axial position
Δz	Length of increment
μ	Dynamic viscosity
ρ	Density

ABBREVIATIONS

1D	One Dimensional
3D	Three Dimensional
CFD	Computational Fluid Dynamics
FNX	Flownex
GT-MHR	Gas turbine-modular helium reactor
HTGR	High Temperature Gas-cooled Reactor
SACAM	South African Conference on Computational and Applied Mechanics
SCFM	Single-Channel Fuel Module
TRISO	Tristructural-isotropic
VHTR	Very High Temperature Reactor

TABLE OF CONTENTS

ACKNOWLEDGEMENTS ..... **I**

PREFACE ..... **II**

NOMENCLATURE ..... **III**

ABBREVIATIONS ..... **IV**

CHAPTER 1: INTRODUCTION **1**

CHAPTER 2: A SYSTEM CFD MODEL OF A SINGLE-CHANNEL FUEL MODULE IN A PRISMATIC BLOCK VHTR **3**

CHAPTER 3: THREE- AND ONE-DIMENSIONAL MODELLING OF A SINGLE-CHANNEL FUEL MODULE IN A PRISMATIC BLOCK VHTR **13**

CHAPTER 4: CONCLUSIONS AND RECOMMENDATIONS **40**

CHAPTER 1: INTRODUCTION

Generation IV nuclear reactor designs include the design for the Very High Temperature Reactor which has a thermal power output of 200–600 MW_{th}. The reactors operate at outlet temperatures of 700 ˚C to 1000 ˚C and at a pressure of 4 MPa to 9 MPa. Graphite is used for the moderator as material and helium is used as the reactor coolant. The fuel compacts consist of Tristructural-isotropic (TRISO) particles dispersed in a graphite matrix. In the gas turbine-modular helium reactor (GT-MHR) prismatic fuel blocks are stacked to make 8000 mm fuel assemblies. A top graphite reflector of 1200 mm and a bottom graphite reflector of 800 mm are added to make the total fuel core height 10000 mm.

The heat transfer and flow through the prismatic fuel blocks are very important for the safe operation of the VHTR core. Computational Fluid Dynamics (CFD) has been used in the industry to simulate different phenomena in the fuel blocks to a very accurate level. A convection heat transfer correlation was developed by Travis & El-Genk (2013b) to calculate the convection heat transfer coefficient for the helium in the coolant channels. In this study the heat transfer and fluid flow were simulated in a representative manner using different CFD codes. The simulations used full 1D, full 3D and a 3D/1D coupled CFD models to model a $1/6^{th}$ Single Channel Fuel Module (SCFM). The 1D simulations were performed using Flownex and the full 3D simulations were executed employing STAR CCM+. The 3D/1D coupled simulations were performed using ANSYS Fluent coupled to Flownex. The simulations account for the conduction heat transfer through the fuel compact and moderator graphite, the convection heat transfer from the coolant channel wall to the coolant channel fluid and the flow of the coolant in the coolant channel.

The results show that due to the mixing that takes place at the entrance of the heated fuel or heated section of the SCFM the local turbulent heat transfer coefficient has an initially high value which rapidly decreases until it reaches a fully developed value at an axial position equal to 25 channel diameters from the entrance. The corresponding variations of the local heat transfer coefficient, the temperature of the coolant channel wall, the bulk fluid temperature and the temperature at the centre of the fuel compact obtained by the different models are compared to determine the validity of the results.

The dissertation comprises of two articles. In chapter 2 the development and evaluation of the full 1D model of the $1/6^{th}$ of the SCFM is discussed. This paper was presented at the 11^{th} South African Conference on Computational and Applied Mechanics (SACAM2018) and is included in the proceedings of SACAM2018. Chapter 3 deals with the development and evaluation of the full 3D and the coupled 3D/1D model. It is intended to submit this paper for publication in an

appropriate journal. A brief overview of the main outcomes and two recommendations are given in chapter 4.

CHAPTER 2: A SYSTEM CFD MODEL OF A SINGLE-CHANNEL FUEL MODULE IN A PRISMATIC BLOCK VHTR

Eleventh South African Conference on Computational and Applied Mechanics
SACAM 2018
Vanderbijlpark, South Africa, 17-19 September 2018

A System CFD Model of a Single-Channel Fuel Module in a Prismatic Block VHTR

GJ Nel[a], CG du Toit[b]

[a,b]North-West University, Potchefstroom, South Africa

Gertt.nel@gmail.com[a],jat.dutoit@nwu.ac.za[b]

Abstract

The conduction heat transfer through the prismatic blocks containing the fuel elements in a Very High Temperature Reactor (VHTR) is of crucial importance for the proper operation of the reactor under normal operating conditions and upset conditions. This paper discusses a system computational fluid dynamics (CFD) model that simulates the heat transfer and fluid flow in a single-channel fuel module of a prismatic block in a representative manner. The model consists of a collection of one-dimensional solid conduction heat transfer, convection heat transfer and pipe elements that are arranged in such a manner to represent the heat transfer and fluid flow in the single-channel fuel module using a network approach. The current model represents one sixth of a single-channel fuel module. The validity of the model was investigated by comparing the temperature distribution in the single-channel fuel module for various scenarios with the corresponding values obtained using a detailed CFD model of the single-channel fuel module. This model requires much less computational resources than the detail CFD and unit cell based models and can form the basis of an integrated model for the entire core.

Keywords: Very High Temperature Reactor (VHTR); System CFD; Conduction heat transfer; Convection heat transfer; Wall Temperature; Heat Transfer Coefficient.

INTRODUCTION

Modelling of heat transfer in the prismatic blocks containing the fuel elements is important in the design and analysis of a VHTR. From the analysis of the heat transfer in a block the temperature distribution can be obtained in the fuel compact and the coolant channel itself. The temperature distribution and heat transfer is very important for the operation of a VHTR. CFD can be used to obtain the essential thermal-fluid characteristics [1].

Simulation of the flow and heat transfer in the cores of prismatic block reactors must account for amongst other the heat transfer in the fuel compacts and graphite matrix, flow in the coolant channels, bypass gaps and cross gaps between the blocks and the heat transfer between the graphite matrix and the fluid. Explicit 3D simulations accounting for all the detail of a $1/12^{th}$ of a fuel assembly were performed by Johnson *et al*. [2], Tak *et al*. [3] and Tung *et al*. [4]. Tung *et al*. [5] considered $1/12^{th}$ of a core and demonstrated the time and computational resources it requires. To lessen the computational burden Tak *et al*. [6-7] and Travis & El-Genk [8-9] developed approaches in which the heat flux in the solids are modeled using a detailed 3D approach and the flow in the coolant channels and the gaps using a 1D approach. This enabled them to simulate $1/6^{th}$ of a core with and without bypass flows. A full core analysis employing a 2D axi-symmetric approach were performed by Rousseau & Greyenstein [10] and Maruyama *et al*. [11-12] using 1D network or system CFD approaches. These, however, do not provide the same detail as the 3D approaches. Models of various levels of complexity have therefore been developed over time, but due to the amount of detail none have succeeded in capturing all the detail in a single 3D, coupled 3D/1D or 1D approach.

In the 3D/1D and 1D models the Nusselt numbers for heat transfer required at the coolant channel wall were obtained using well known correlations for turbulent forced convection published in literature [13-14]. Travis & El-Genk [8] performed a 3D and a 3D/1D simulation of a single channel fuel module (SCFM). The purpose was to determine the difference in the computational requirements between the 3D and 3D/1D approaches. The notion was that little accuracy would be sacrificed if the conduction heat transfer through the fuel compacts and graphite would be modelled in 3D detail, but the flow in the coolant channel and the heat transfer at the wall using a 1D methodology. As part of the study they also used the detail from the full 3D simulation to extract the relevant data to determine the Nusselt numbers for the heat transfer at the channel wall and compared the results with those predicted by correlations. Travis & EL-Genk [8] proposed a new correlation and then applied it in the simulation of the $1/6^{th}$ core [9].

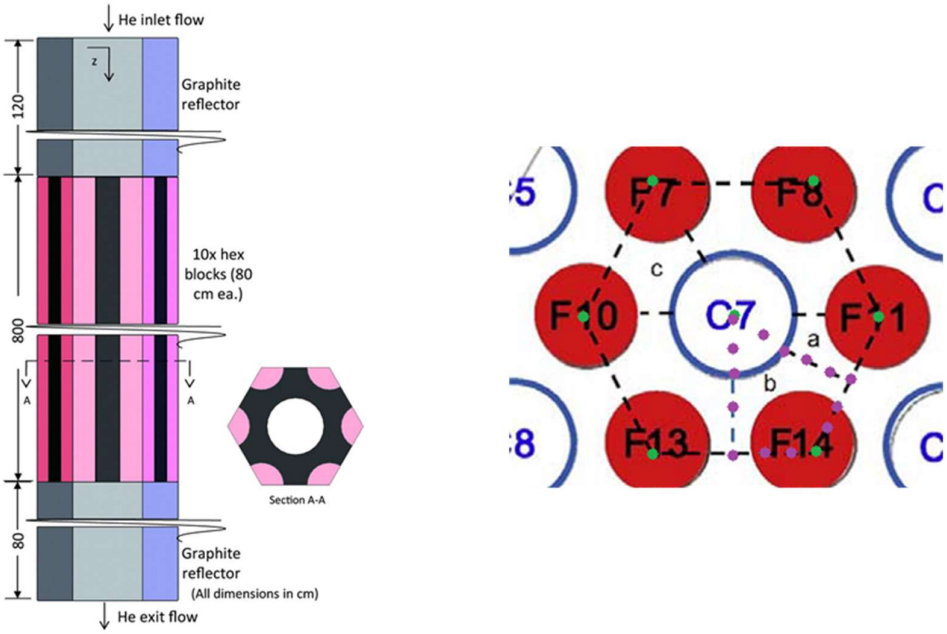

Fig. 1: (a) A hexagonal single-channel fuel module for a VHTR core (b) SCFM

In the current study a Flownex system or network model was developed of the SCFM incorporating the Nusselt number correlation proposed by Travis & El-Genk [9]. Only 1/6th of the SCFM was modelled due to the symmetry in the geometry. Heat transfer through the graphite was modelled using a conduction shape factor. In the study the fluid and coolant channel wall temperatures as well as the heat transfer coefficients predicted by the Flownex model are compared with the Travis & El-Genk [9] results.

SINGLE CHANNEL FLOW MODULE

Travis & El-Genk [8] studied temperature distribution, heat flux and flow through a SCFM using a full 3D approach and a combined 3D/1D approach. Fig. 1 (a) and (b) show a SCFM of a prismatic core block where the red elements represent the fuel compacts and the blue circle elements represent the coolant channels. Each of these blocks are 800 mm in height and ten of these blocks are stacked to form a fuel block assembly which is 8000 mm in length. Within the partial core block a single channel fuel module consists of one coolant channel and six partial fuel compacts. The SCFM is represented by the area contained around the coolant channel outlined by the green dots in Fig.1 (b).

The different materials used in the model have material properties that are represented by the correlations given in Table 1.

Table 1. Properties of materials used in SCFM of a VHTR

Property	Material	Correlation
Density, ρ (kg/m^3)	IG-110 graphite	1740
Specific heat, c_p (J/kg.K)		$6.05*10^{-7}T^3$ -0.00269T^2 +4.19T -294
Thermal conductivity, k (W/m.K)		$-13.2 +2.50*10^4/(T+268)^{0.78}$
Density, ρ (kg/m^3)	Composite Fuel	1650
Specific heat, c_p (J/kg.K)		$3.11*10^{-7}$ T^3 -0.00155 T^2 +2.73 T -82.4
Thermal conductivity, k (W/m.K)		$8.5 +7.68*10^4/(T+268)^{0.995}$
Density, ρ (kg/m^3)	Helium	P/R_gT
Specific heat, c_p (J/kg.K)		5197.6
Thermal conductivity, k (W/m.K)		0.000258 T +0.103388
Dynamic viscosity, μ (Pa-s)		0.03319 T +13.0744

Mass flow rate, inlet pressure and inlet temperature are used as boundary conditions. The mass flow rate is set at 0.0306 kg/s and the inlet pressure at 7.07 MPa, whilst the inlet temperature is set at 914 K. The total power applied to the SCFM is 55.4 kW which corresponds to a 600 MW_{th} HTGR. Two power profiles were assumed namely a uniform distribution and a (chopped) cosine distribution. In the case of the uniform distribution a power density of 27.4026 MW_{th}/m^3 [8] was prescribed. In case of the cosine power profile the power density $PD(z)$ at the distance z from the entrance of the coolant channel is given as:

$$PD(z) = PD_{max} * \cos(0.2566 * z - 0.8831) \quad (1)$$

Where the maximum power density is given as PD_{max}=33.135 MW_{th}/m^3.

Star CCM+ was used by Travis & El-Genk [8] to simulate the 3D model. A very fine mesh was used for the axial and the radial meshing of the 3D model at the common interfaces. Thus a large number of the 6.67 million cells in the mesh were situated at the area where the fuel compact and the graphite block interface, as well as at the interface between the graphite wall and the coolant channel where the convection heat transfer occurs [8]. The

same simulation in Star CCM+ was performed with 4.67 million cells and the results showed a difference of less than 0.1%. From the results the wall temperatures, T_w, and the bulk fluid temperatures, T_b, as well as the heat fluxes at the wall surface of the coolant channel were extracted. Travis & El-Genk [8] used these values to calculate the local Nusselt number.

From the result Travis & El-Genk [8] derived a correlation for the turbulent Nusselt number Nu_{FD} for fully developed flow for $z/D \geq 25$ given as:

$$Nu_{FD} = 0.11 Re_b^{0.646} Pr_b^{0.4} \quad (2)$$

with D the diameter of the coolant channel. Re_b and Pr_b are the Reynolds and Prandtl numbers based on the bulk temperature of the fluid. For $z/D < 25$ the correlation was adapted by Travis & El-Genk [8] to account for developing flow. The adapted correlation is given as:

$$Nu = Nu_{FD} * (1 + 0.57 e^{\frac{-0.2z}{D}}) \quad (3)$$

The local heat transfer coefficient of the helium in the coolant channel was calculated for the 3D and 1D approaches by Travis & El-Genk [8]. The comparison between the heat transfer coefficients as a function of the normalized distance obtained by Travis &El-Genk [8] for the cosine power profile is shown in Fig.2. It can be seen that the agreement is very good. Also shown in Fig.2 is the comparison between the bulk fluid temperatures obtained by the two approaches.

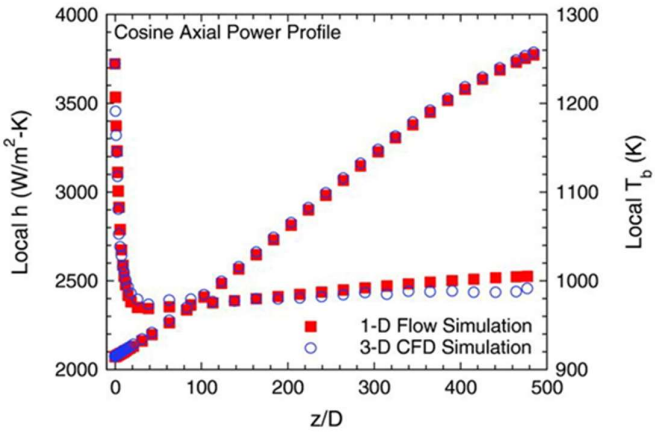

Fig. 2: Comparison of calculated values of the local heat transfer coefficient using 1-D and 3-D simulation of helium gas flow in the SCFM

Travis & El-Genk [9] subsequently applied the 3D/1D methodology to simulate the heat transfer and flow in 1/6th of a full height prismatic core and concluded that the results confirmed the effectiveness of the methodology.

FLOWNEX MODEL

Flownex is a system level simulation tool for the modeling of thermal-fluid systems and is used across multiple industries in the design and optimization of fluid systems. Flownex calculates amongst others the flow rates, temperatures, pressures and heat transfer rates using both steady state and transient models [15].

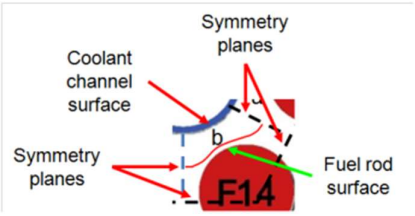

Fig. 3: Unit cell for conduction between fuel rod and coolant channel.

In the Flownex model only 1/6th of the SCFM as shown in Fig.3 is considered due to the symmetry that can be observed in Fig.1 (b). The model therefore consists of 1/6th of a coolant channel, 1/3rd of a fuel compact and the associated graphite matrix or moderator. The conduction heat transfer through the fuel compact is modelled using one conduction component as shown in Fig.4. The conduction heat transfer through the graphite from the surface of the fuel rod to the surface of the coolant channel is modelled using two conduction components. A convection component is used to model the convection heat transfer between the wall of the coolant channel and the helium in the coolant channel. The convective heat transfer and flow in the coolant channel is modelled using a pipe component. The appropriate conduction lengths and areas for the graphite conduction components were obtained from the shape factors calculated by Sambureni [16]. The material properties given in Table 1 were implemented in the Flownex model using material property tables. The correlations (2) and (3) for the turbulent forced convection Nusselt number were implemented through the appropriate user coding in the convection component.

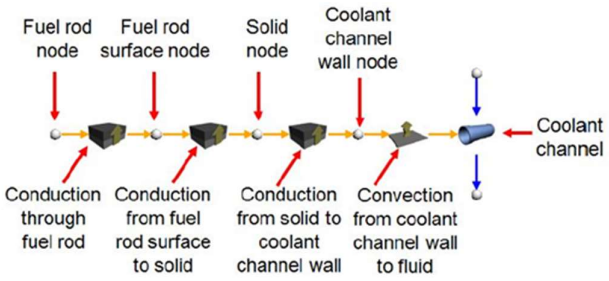

Fig. 4: Flownex network for heat transfer from fuel rod and coolant channel

The SCFM is discretized into 10 primary increments each with a length of $\Delta z = 0.8\ m$ as shown in Fig.5. To account for the inlet effects three discretization strategies were tested. In two cases the first increments (top increment in Fig.5) was divided into two increments of 0.4 m each. The first of the two increments corresponding to $z/D \leq 25$ was then subsequently subdivided into five or ten increments with lengths of 0.08 m and 0.04 m respectively.

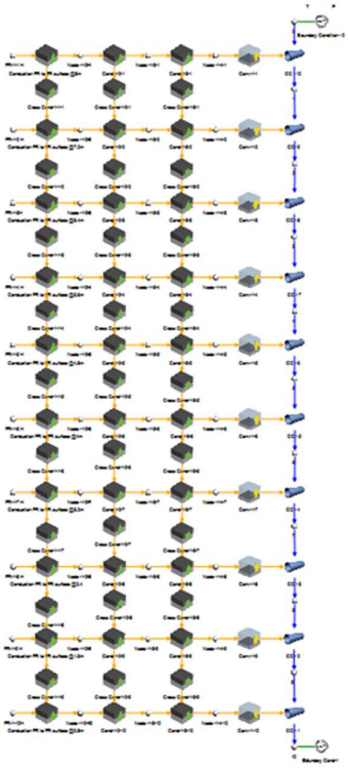

Fig. 5: 800 mm x 10 increment Flownex model

The total power input for the 1/6th SCFM is 9.233 kW_{th}. When a uniform power profile is considered the power input $P(z)$ for an increment Δz *is* given as:

$$P(z) = PD_{max}.A.\Delta z \tag{3}$$

with $PD_{max} = 27.403\ MW_{th}/m^3$ and A the cross sectional area of the 1/3rd fuel compact. In the case of the cosine power profile the power input for an increment Δz *is* given as:

$$P(z) = PD_{max}.A.[2.47373\sin(0.2566z) - 3.01134\cos(0.2566z)]_{z-\frac{1}{2}\Delta z}^{z+\frac{1}{2}\Delta z} \tag{4}$$

where $PD_{max} = 33.135\ MW_{th}/m^3$. The axial position z is taken in the middle of the increment. The power for an increment is distributed to the two fuel nodes associated with the increment according to the volumes of the control volumes represented by the nodes. This leads to 25 % of the power allocated to the first node and 75 % of the power allocated to the surface node.

RESULTS

In the Flownex model the pressure and temperature at the inlet boundary (top node in Fig.5) were specified as 7.07 MPa and 914 K respectively. The mass flow rate exiting at the outlet boundary (bottom node in Fig.5) was specified as 0.0051 kg/s. The wall temperature, bulk temperature and the heat transfer coefficients were extracted from the Flownex results and are compared with the corresponding values obtained by Travis & El-Genk [8].

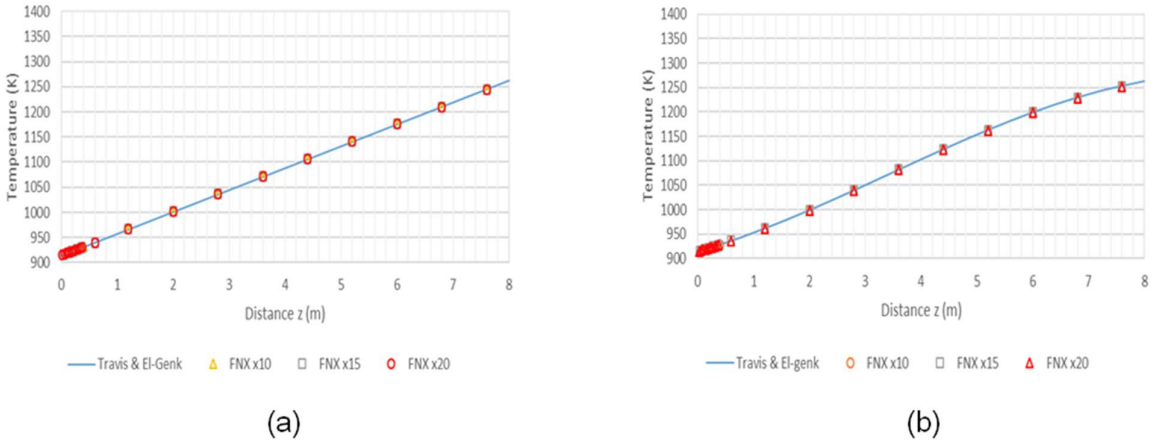

Fig. 6: Fluid Temperatures as function of axial position, (a) Uniform power profile (b) Cosine power profile.

In Fig.6 (a) the comparison between the axial variation of the temperatures obtained by the Flownex models and Travis & El-Genk [8] for the uniform power profile is shown, whilst the corresponding comparison for the cosine power profile is shown in Fig.6 (b). The agreement between the various sets of results is very good.

The comparison between the axial variation of the wall temperatures obtained by the Flownex model and Travis & El-Genk [8] for the uniform power profile is shown in Fig.7 (a), whilst in Fig.7 (b) the corresponding comparison for the cosine power profile is shown. Again the agreement between the various sets of results is very good.

Fig.8 (a) shows the comparison between the local values of the heat transfer coefficient as a function of the axial position as predicted by the Flownex models and obtained by Travis & El-Genk [8] for the uniform power profile. The corresponding comparison for the cosine power profile is shown in Fig.8 (b). In this case the various sets of results are in close agreement.

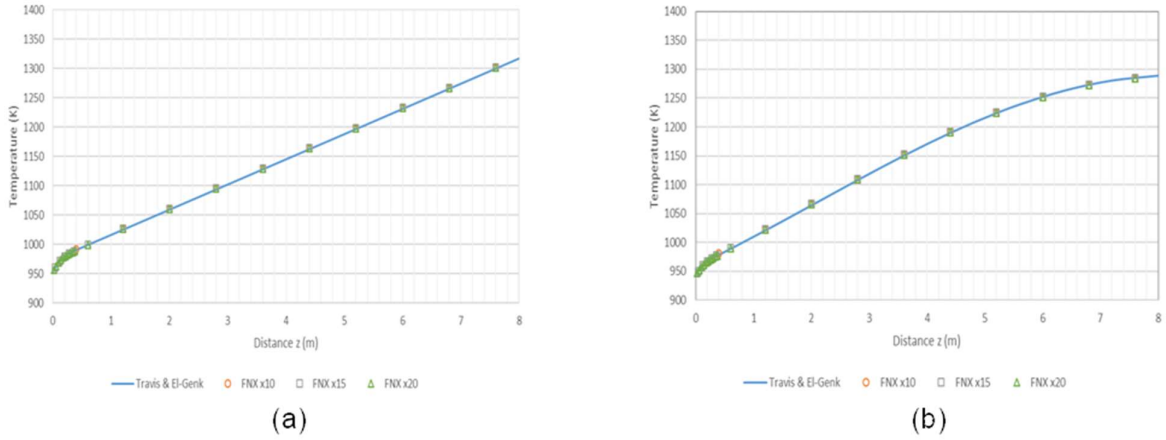

Fig. 7: Wall Temperatures as function of axial position (a) Uniform power profile (b) Cosine power profile.

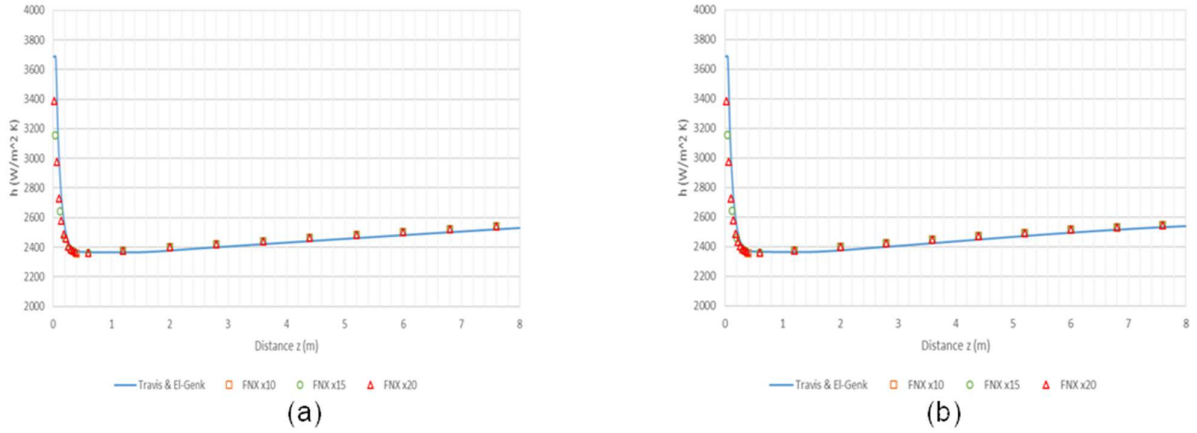

Fig. 8: Local heat transfer coefficients as function of axial position (a) Uniform power profile (b) Cosine power profile.

To complete the comparative study the well-known turbulent forced convection Dittus-Boelter Nusselt number correlation [13]:

$$Nu = 0.023 Re_b^{0.8} Pr_b^{0.4} \left(\frac{T_w}{T_b}\right)^{-0.5} \tag{5}$$

incorporated in Flownex, as well as the corresponding correlation by McEligot *et al*. [14]:

$$Nu = 0.021 Re_b^{0.8} Pr_b^{0.4} \left(\frac{T_w}{T_b}\right)^{-0.5} \left[1 + \left(\frac{z}{D}\right)^{-0.7}\right] \tag{6}$$

incorporated in Flownex in this study, were also used to predict the heat transfer coefficients. A comparison between the local values of the heat transfer coefficients predicted by the various Flownex implementations and Travis & El-Genk [8] value is shown in Fig.9. In all cases the Flownex model consisting of 20 increments was employed. The values for the Dittus-Boelter correlation do not capture the inlet effect and differ an average by 7.24 % from the Travis & El-Genk [8] values in the fully developed region. The values for the McEligot *et al*. [14] correlation take a longer distance to reach a fully developed state, but are in very good agreement with Travis & El-Genk [8] values after approximately 3.5 m.

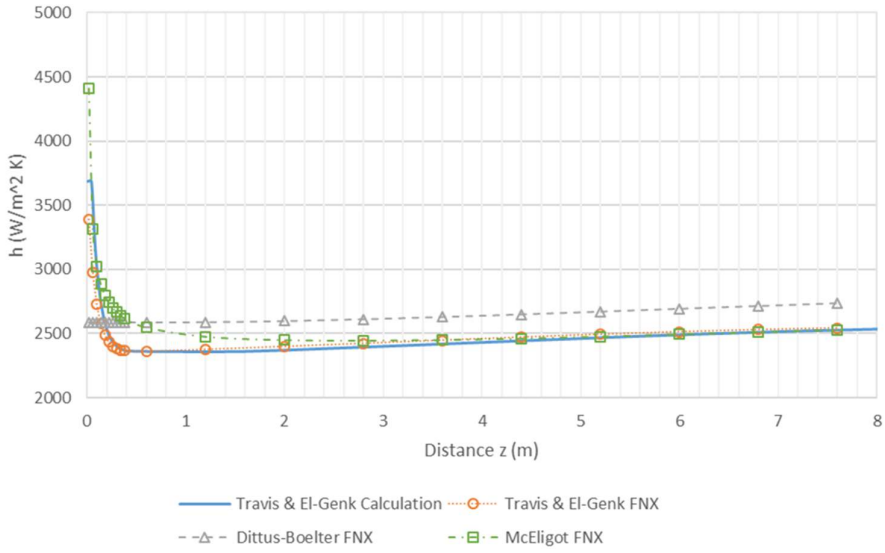

Fig. 9: Local heat transfer coefficients as function of axial position for selection correlations (x20 Increments).

It can therefore be concluded that the Travis & El-Genk [8] correlation for the turbulent forced convection Nusselt number have been successfully implemented in Flownex and that the system model is capable of predicting the temperatures accurately.

CONCLUSION

In this paper the development of a Flownex model of 1/6th of a single-channel fuel module (SCFM) of a prismatic fuel block of a VHTR was discussed. The study is based on work performed by Travis & El-Genk [8] and their Nusselt number correlation for turbulent forced convection was implemented in the Flownex model. A uniform power profile and cosine power profile were considered in the study. It was found that the values predicted by the Flownex models for the wall temperature, bulk fluid temperature and the local heat transfer coefficient were in very good agreement with corresponding values obtained by Travis & El-Genk [8]. It was concluded that the Travis & El-Genk [8] correlation for turbulent forced convection was implemented successfully in Flownex and that a Flownex system model is capable of predicting the wall and fluid temperatures for the SCFM accurately.

This study is the first step to investigate the possibility to couple a Flownex 1D model of the flow in a coolant channel to a 3D model of the heat transfer through the fuel compacts and graphite matrix of a SCFM be investigated.

Acknowledgements

This work is based on the research supported by the South African Research Chairs Initiative of the Department of Science and Technology and the National Research Foundation of South Africa (Grant No 61059).

Any opinion, findings and conclusion or recommendation expressed in this material is that of the authors and the NRF does not accept any liability in this regard.

1. References

[1] Du Toit, CG, Rousseau, PG, Sambureni, P, Le Grange, LA (2015). *A System CFD Model of the Heat Transfer in a Prismatic Block of a VHTR*, *Proceedings of the 2015 International Congress on Advances in Nuclear Power Plants*, Nice, France, paper 15167.

[2] Johnson,RW, Sato, H. (2012). Bypass flow computations using a one-twelfth symmetric sector for normal operationin a 350 MWth prismatic VHTR. *Nuclear Engineering and Design,* vol. 251, p. 84–91

[3] Tak, NI, Kim, MH, Lee, WJ (2008). Numerical investigation of a heat transfer within the prismatic fuel assembly of a very high temperature reactor. *Annals of Nuclear Energy,* vol. 35, p. 1892–1899

[4] Tung, YH, Ferng, YM, Johnson, RW, Chieng, CC (2013). Study of natural circulation in a VHTR after a LOFA using different turbulence models. *Nuclear Engineering and Design,* vol. 263, p. 206– 217

[5] Tung, YH, Ferng, YM, Johnson, RW, Chieng, CC (2016). Transient LOFA computations for a VHTR using one-twelfth core flow models. *Nuclear Engineering and Design,* vol. 301, p. 89–100

[6] Tak, NI, Kim, MH, Lim, HS, Noh, JM (2012). A practical method for whole-core thermal analysis of a prismatic gas-cooled reactor. *Nuclear Technology,* vol. 177, pp. 352-365

[7] Tak, NI, Lee, SN, Kim, MH, Lim, HS, Noh, JM (2014). Development of a core thermo-fluid analysis code for prismatic gas cooled reactors. *Nuclear Engineering and Technology,* vol. 46, no. 5, pp. 641-654

[8] Travis, BW, El-Genk, MS (2013). Numerical Simulation and Turbulent Convection Heat Transfer Correlation for Coolant Channels in a Very-High-Temperature Reactor. *Heat Transfer Engineering*, 34, no 1, pp. 1–14

[9] Travis, BW, El-Genk, MS (2013). Thermal–hydraulics analyses for 1/6 prismatic VHTR core and fuel element with and without bypass flow. *Energy Conversion and Management,* vol. 67, p. 325–341

[10] Rousseau, P. G., Greyvenstein, GP (2002). Modelling of the HTTR in Flownex. *Proceedings of the 1st International Topical Meeting on High Temperature Reactor Technology*, Petten, Netherlands.

[11] Maruyama, S, Yamashita, K, Fujimoto, N, Murata, I, Sudo, Y (1993). Evaluation of Hot Spot Factors for Thermal and Hydraulic Design of HTTR. *Journal of Nuclear Science and Technology,* vol. 30, no. 11, pp. 1186-1194

[12] Maruyama, S, Fujimoto, N, Sudo, Y, Murakami, T, Fujii, S (1994). Evaluation of core thermal and hydraulic characteristics of HTTR. *Nuclear Engineering and Design,* vol. 152, pp. 183-196

[13] Incropera, FP, DeWitt, DP (1996). *Fundamentals of Heat and Mass Transfer*. 4th edition, John Wiley & Sons

[14] McEligot, DM, McCreery, GE, Schultz, RR, Lee, J, Hejzlar, P, Stahle, P, Saha (2006). *Investigation of Fundamental Thermal-Hydraulic Phenomena in Advanced Gas-Cooled Reactors*, INL/EXT-06-11801, MIT-GFR-042.

[15] M-Tech Industrial (2017) *Flownex SE Version, 8.8.1.3243*, http://www.flownex.com

[16] Sambureni, P (2015). *Thermal fluid network model for a prismatic block in a gas-cooled reactor using FLOWNEX*, Masters dissertation, North-West University.

CHAPTER 3: THREE- AND ONE-DIMENSIONAL MODELLING OF A SINGLE-CHANNEL FUEL MODULE IN A PRISMATIC BLOCK VHTR

Three- and one-dimensional modelling of a Single-Channel Fuel Module in a Prismatic Block VHTR

GJ Nel[a], CG du Toit[b]

[a,b]*North-West University, Potchefstroom, South Africa*

Gertt.nel@gmail.com[a],jat.dutoit@nwu.ac.za[b]

Abstract

The conduction heat transfer through the prismatic blocks containing the fuel elements in a Very High Temperature Reactor (VHTR) is of crucial importance for the proper operation of the reactor under normal operating conditions and upset conditions. This paper discusses 3D, 1D and 3D/1D coupled computational fluid dynamics (CFD) models that simulate the heat transfer and fluid flow in a single-channel fuel module of a prismatic block in a representative manner. These models account for conduction heat transfer in the solids, convection heat transfer between the solids and fluid and fluid flow in the coolant channel in the single-channel fuel module. The current model represents one sixth of a single-channel fuel module. The validity of the models are investigated by comparing the temperature distribution in the single-channel fuel module for various scenarios with the corresponding values obtained using a detailed CFD model of the single-channel fuel module. The 1D and 3D/1D models require much less computational resources than the detail 3D CFD and can form the basis of an integrated model for the entire core.

Keywords: Prismatic fuel block; heat transfer; conduction; convection; 3D modeling; 1D modeling.

Introduction

The Very High Temperature Reactors (VHTR) are generation IV designs which can deliver thermal power of up to 600 $MW_{th.}$ Graphite is used as the material for the moderator and the reactor is helium cooled allowing outlet temperatures that can range from 700-1000 ˚C with operating pressures of 4 MPa to 9 MPa (Travis & El-Genk, 2013b). In the core of the Prismatic VHTR the hexagonal fuel blocks are stacked in fuel assemblies with a height of 8000 mm and with graphite reflector blocks at the bottom and the top of the fuel assemblies. A fuel block typically contains 102 coolant channels and 210 fuel holes filled with fuel rods. The fuel compacts that make up a fuel rod are made of TRISO (Tristructural-isotropic) particles packed in a graphite matrix. One single coolant channel is surrounded by six fuel channels and this pattern is repeated throughout the hexagonal blocks in the core.

Numerical models of prismatic block reactors must amongst others account for the thermal-fluid phenomena in the fuel compacts, graphite, coolant channels in the fuel blocks, bypass and cross gaps between blocks and the mixing in the upper and lower plenums. The extensive amount of detail that must be considered when modelling a full reactor cannot be accounted for a single explicit 3D numerical model. Contributions that account only for selected aspects in explicit 3D detail; contributions that use various combinations of explicit 3D detail, implicit 3D detail, implicit 1D approximations and systems or network formulations to address larger portions of the reactor; and systems formulations that can simulate a full reactor unit are therefore reported in literature.

Johnson *et al*. (2012), Tak *et al*. (2008), and Tung *et al*. (2013, 2016) modelled $1/12^{th}$ of a fuel assembly with explicit detail of the fuel compacts, graphite, coolant channels and bypass gaps. They studied the effect of prescribed power profiles and bypass gap size on the thermal-flow behaviour of the fuel assemblies. Tung *et al*. (2013, 2016) also studied the natural circulation that occurs after a loss of flow accident (LOFA). Johnson *et al*. (2012) used the CFD code ANSYS Fluent, Tak *et al*. (2008) the CFD code CFX, whilst Tung *et al*. (2013, 2016) used the CFD code STAR CCM+.

Yoon *et al*. (2012) and Lee *et al*. (2016) used explicit 3D detail to simulate the flow in the coolant channels, bypass gaps and cross gaps of an iso-thermal fuel block assembly. Yoon *et al*. (2012) studied the effect of variable bypass gap sizes, whilst Lee *et al*. (2016) characterized the pressure over a cross gap to obtain a correlation that could be implemented in a 1D network code. Both Yoon *et al*. (2012) and Lee *et al*. (2016) used the CFD code CFX and compared their results with measurements obtained from experiments.

Travis and El-Genk (2013a) studied the thermal-flow in a single fuel block and a single control block with no bypass flow and bypass flow using explicit 3D detail for the solid conduction and

the flow paths. They compared the results with simulations where the solid conduction was modelled in 3D detail and the flow paths using a 1D approach. They found the agreement to be very good. They subsequently modelled 1/6th of a core using 3D detail for the solid conduction and the 1D approach for the coolant channels and bypass gaps and control rod flow. For the 3D detail they employed the CFD code STAR CCM+.

Tak *et al*. (2012) and Tak *et al*. (2014) studied the heat transfer and flow in standard, control and reserve shut down fuel assemblies using 3D explicit detail for the heat transfer in the solids and a 1D formulation for the flow paths for various power profiles using the code CORONA. The results were found to be in good agreement with results obtained from 3D explicit detail simulations using the CFD code CFX. They also used the 3D/1D approach employed in CORONA to model the thermal-flow in a 1/6th sector of a core.

The effect of coolant channel blockages was studied by Lee *et al*. (2014) considering a 1/12th fuel assembly using CORONA and compared the 3D/1D results with those obtained from a 3D explicit detail simulation using the CFD code CFX. The agreement was good and they therefore used CORONA and studied the effect on the flow and heat transfer in 1/6th of a core of single and multiple coolant channel blockages.

Cioni *et al*. (2006) used the CFD code Trio_U to model the effect that the blockage of selected coolant channels have on the heat transfer and flow in a fuel assembly surrounded by partial fuel assemblies. The heat transfer in the solids was modelled employing 3D explicit detail, whilst the flow in the coolant channels and bypass gaps were modelled using a 1D approach.

A study on the effect of various bypass gap and cross gap sizes on the thermal-flow in 1/6th of core was performed by Kim and Lim (2011) using the 1D / 2D / 3D system code GAMMA+. The heat transfer and flow in the fuel blocks were modelled implicitly and a 1D approach was used for the flow in the bypass and cross gaps.

Maruyama *et al*. (1993), Maruyama *et al*. (1994) simulated the thermal-flow behaviour of the high temperature test reactor (HTTR) using the system code FLOWNET to model to flow in the coolant channels, bypass gaps and cross gaps. The 2D axi-symmetric network code TEMDIM was used to calculate the heat transfer in the fuel compacts and graphite. Rousseau and Greyvenstein (2002) also employed a 2D axi-symmetric network model using the system code Flownex to simulate the thermal-flow behaviour of the HTTR. They, however, only considered the heat transfer through the fuel and reflector blocks and the flow in the coolant channels.

In the coupling between the 3D explicit modelling of the heat transfer in the solids and the 1D modelling of the flow in the coolant channels it is important to use an appropriate convection heat transfer correlation. Travis and El-Genk (2013b) performed a 3D explicit detail simulation of a single channel fuel module (SCFM) using the CFD code STAR CCM+ to evaluate various

convection heat transfer correlations. They then proposed an improved convection heat transfer correlation and implemented that in a coupled 3D/1D simulation of the SCFM. The 3D heat transfer was modelled using STAR CCM+. No information is given on the 1D formulation that they employed.

In none of the coupled 3D/1D simulations reported in the literature any description is given of the way in which the convection heat transfer coupling between the 3D heat transfer and the 1D fluid flow was implemented.

In this study models were considered that represent 1/6th of the cross section of the SCFM considered by Travis and El-Genk (2013b). Nel and Du Toit (2018) first constructed a 1D model of the fuel compact, graphite moderator and coolant channel using the system CFD code Flownex. They implemented the convection heat transfer coefficient correlation proposed by Travis and El-Genk (2013b) and found the coolant channel wall temperatures to be in good agreement with the corresponding temperatures obtained from Travis and El-Genk (2013b). An explicit 3D model of the fuel compacts, graphite moderator and coolant channel of the 1/6th of the SCFM was then created using the CFD code STAR CCM+. The calculated heat flux at the coolant channel wall, the wall temperatures and the bulk fluid temperatures were then used to obtain the local convection heat transfer coefficient. These values were found to be in good agreement with the corresponding values predicted by the Travis and EL-Genk correlation. Lastly a coupled 3D/1D model of the 1/6th SCFM was generated. ANSYS Fluent was used for the explicit 3D representation of the fuel compact and graphite moderator and the system CFD Flownex was used for the 1D representation of the coolant channel. The Travis and El-Genk convection heat transfer correlation was employed to couple the heat transfer between the 3D and 1D formulations. The results for the coolant channel wall temperatures and fuel compact centre temperatures were found to be in very good agreement with corresponding values predicted by the full 3D explicit model.

Single-Channel Fuel Module

Within a fuel block a single channel fuel module consists of one coolant channel and six partial fuel compacts. Fig. 1 (a) and (b) show a SCFM of a prismatic core block where the red elements in Fig 1(b) represent the fuel compacts and the blue circle elements represent the coolant channels. Ten fuel blocks each with a height of 800 mm are stacked to form a fuel block assembly which has a total height of 8000 mm. The SCFM is represented by the hexagonal cross sectional area contained around the coolant channel in Fig. 1(a) and the area outlined by the dashed hexagon in Fig.1 (b). The diameters of the fuel compacts and the coolant channels are respectively 12.7 mm and 15.875 mm and the distance from a fuel compact center to the opposite fuel compact centre is 37.6 mm.

The temperature distribution, heat flux and flow through a SCFM was studied by Travis & El-Genk (2013b) using a 3D CFD approach.

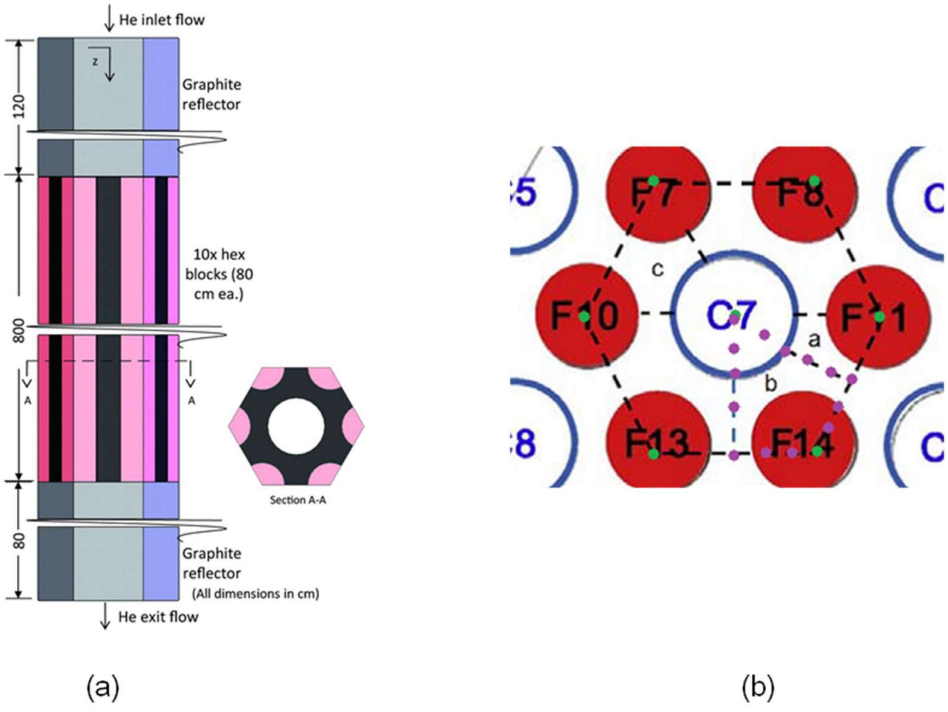

Fig. 1: (a) Side and top view of the SCFM; and (b) dashed hexagon outlining SCFM cross sectional area.

Only a section of the SCFM needs to be modeled based on the symmetry that can be observed in Fig. 1(b). Three unit cells that can be selected are indicated in Fig. 1(b) by the dashes areas a, b, and c. Unit cell a represents 1/12th of the SCFM, whilst unit cells b and c represent 1/6th of the SCFM. Unit cell b was selected in this study due to the fact that Sambureni (2015) had characterized the conduction shape factor for the unit cell.

The mass flow rate, inlet pressure and inlet temperature were used as boundary conditions. The mass flow rate was assumed to be 0.0306 kg/s and the inlet pressure to be 7.07 MPa, whilst the inlet temperature was assumed to be 914 K. The total power applied to the SCFM was 55.4 kW which corresponds to a 600 MW_{th} HTGR. Two power profiles were assumed namely a uniform distribution and a (chopped) cosine distribution. In the case of the uniform distribution a power density of 27.4026 MW_{th}/m^3 (Travis & El-Genk, 2013b) was prescribed. In case of the cosine power profile the power density *PD(z)* at the distance z from the entrance of the coolant channel is given as:

$$PD(z) = PD_{max} * \cos(0.2566 * z - 0.8831) \quad (1)$$

Where the maximum power density is given as PD_{max}=33.135 MW_{th}/m^3.

The properties of the different materials are given by the correlations summarized in Table 1.

Table 1. Properties of materials used in SCFM (Travis & El-Genk, 2013b).

Property	Material	Correlation
Density, ρ (kg/m^3)	IG-110 graphite	1740
Specific heat, c_p (J/kg.K)		$6.05*10^{-7}T^3$ -0.00269T^2 +4.19T -294
Thermal conductivity, k (W/m.K)		$-13.2 +2.50*10^4/(T+268)^{0.78}$
Density, ρ (kg/m^3)	Composite Fuel	1650
Specific heat, c_p (J/kg.K)		$3.11*10^{-7}$ T^3 -0.00155 T^2 +2.73 T -82.4
Thermal conductivity, k (W/m.K)		$8.5 +7.68*10^4/(T+268)^{0.995}$
Density, ρ (kg/m^3)	Helium	P/R_gT
Specific heat, c_p (J/kg.K)		5197.6
Thermal conductivity, k (W/m.K)		0.000258 T +0.103388
Dynamic viscosity, μ (Pa-s)		0.03319 T +13.0744

Based on the result obtained by Travis and El-Genk (2013b) it was found that for the conditions given we have that $4.46\times10^4 < \mathrm{Re} < 5.65\times10^4$ and $1.28\times10^3 < Ra < 7.92\times10^3$. These result in an Aicher number of $Ai < 0.05$ and the convection heat transfer regime can therefore be considered as turbulent forced convection heat transfer as shown in Fig. 2.

Travis & El-Genk

In the study conducted by Travis & El-Genk (2013b) they developed a turbulent convection heat transfer correlation from the results of a full 3D CFD numerical model of the heat transfer and flow in the SCFM. The analysis accounted for material property changes of the moderator graphite, coolant fluid (Helium) and the fuel compacts as a function of the temperature variation in the SCFM. They considered two different power profiles, namely a uniform profile and a chopped-cosine profile, and the analysis therefore also accounted for the effect that different power distributions might have.

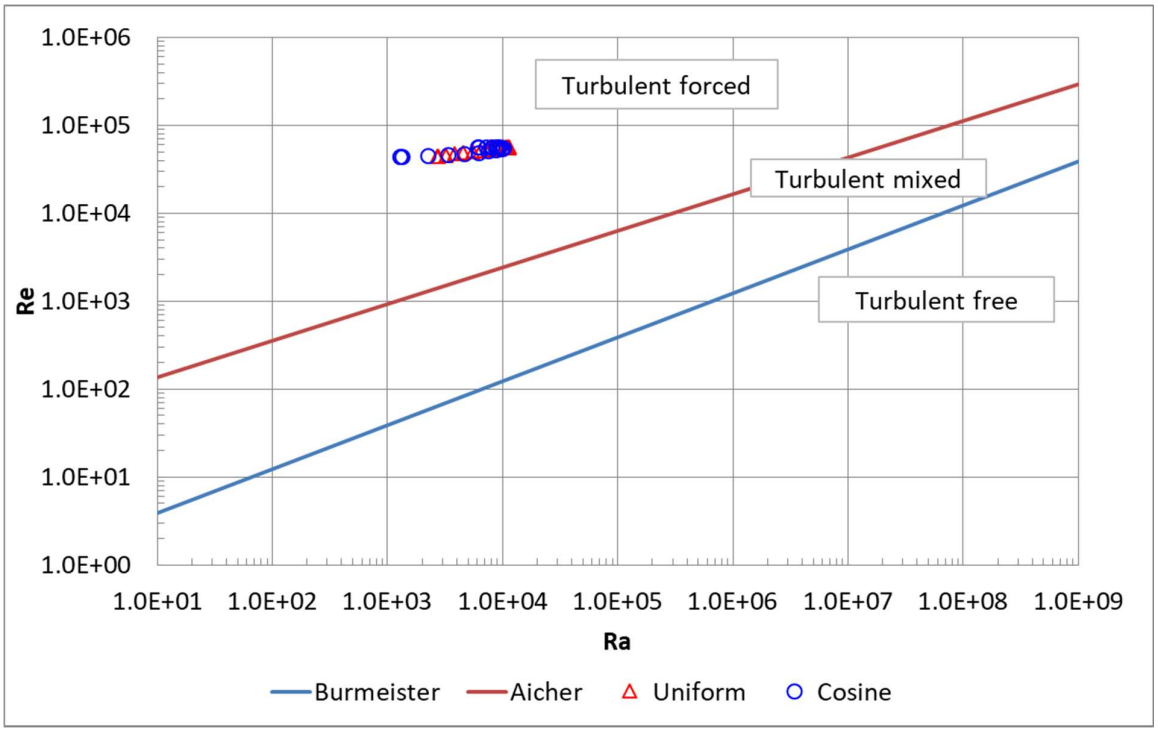

Fig. 2: Convection heat transfer regime for SCFM.

Travis & El-Genk (2013b) used the CFD code STAR CCM+ to simulate the heat transfer and fluid flow in the 3D explicit model. Prism layer meshes were used at the interfaces between the fuel compact and moderator graphite, and at the interface between the moderator graphite and the coolant. The final mesh that was employed in the simulations consisted of 6.67 million cells. The corresponding results obtain on a mesh consisting of 4.67 million cells differed by less than 0.1 % from those obtained on the fine mesh.

From the results the wall temperatures, T_w, and the bulk fluid temperatures, T_b, as well as the heat fluxes at the wall surface of the coolant channel were extracted. Travis & El-Genk (2013b) used these values to calculate the local heat transfer coefficient. They found that in the first 25

channel diameters after the entrance of the heated section the value of the local heat transfer coefficient decreased sharply before increasing gradually over the remainder of the heated section of the SCFM.

From the result Travis & El-Genk (2013b) derived a correlation for the turbulent Nusselt number Nu_{FD} for fully developed flow for $z/D \geq 25$ given as:

$$Nu_{FD} = 0.11 Re_b^{0.646} Pr_b^{0.4} \quad (2)$$

with D the diameter of the coolant channel. Re_b and Pr_b are the Reynolds and Prandtl numbers based on the bulk temperature of the fluid. For $z/D < 25$ the correlation was adapted by Travis & El-Genk (2013b) to account for developing flow. The adapted correlation is given as:

$$Nu = Nu_{FD} * (1 + 0.57 e^{\frac{-0.2z}{D}}) \quad (3)$$

The values of the Nusselt numbers predicted by eqns. (2) and (3) are within 2 % of the corresponding values obtained from the 3D simulations.

Travis & El-Genk (2013b) then implemented the proposed correlation in a coupled 3D/1D simulation using STAR CCM+ to model the 3D heat transfer in the fuel compact and the moderator graphite and a 1D approach for the flow in the coolant channel. The comparison between the heat transfer coefficients as a function of the normalized distance obtained by Travis & El-Genk (2013b) for the cosine power profile is shown in Fig. 3. It can be seen that the agreement between the results obtained by the full 3D simulation and the coupled 3D/1D simulation is very good. Also shown in Fig. 3 is the comparison between the bulk fluid temperatures obtained by the two approaches. The corresponding values are in very good agreement.

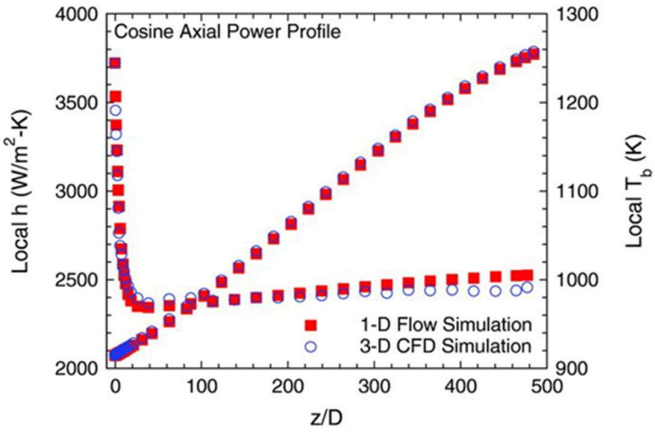

Fig. 3: Comparison of the values of the local heat transfer coefficient and the bulk fluid temperature obtained using coupled 3D/1D and 3-D simulations of the SCFM (Travis & El-Genk, 2013b)

Travis & El-Genk (2013b) subsequently applied the 3D/1D methodology to simulate the heat transfer and flow in 1/6th of a full height prismatic core and concluded that the results confirmed the effectiveness of the methodology.

3D STAR CCM+ Model

A full 3D model of the SCFM was constructed and used in a simulation in STAR CCM+ (SIEMENS, 2018) to recreate the results that were obtained by Travis & El-Genk (2013b). The geometry of the SCFM was firstly generated in Solidworks 2016 and consisted of a 1200 mm long section of the 1/6th SCFM. The assembly was secondly then imported into STAR CCM+ where the necessary parts and surfaces were split and renamed as needed. Different regions and different material continua were then assigned to the different parts and faces. With the three different material continua, namely fuel rod, moderator graphite and helium, which exist the material properties given in Table 1, were assigned to the relevant regions.

A single mesh continua was constructed for all the components in the model so ensure that the cells faces on the interface between adjacent parts matched to form a conformal mesh. This ensured that no interpolation was necessary for the transfer of information across interfaces. It was found to play an important role in the stability of the solution. A basic base size of 6 mm, based on the radius of the fuel compact, was chosen for the mesh. A number of five prism layers was been selected and the layer thickness of all the layers was chosen such that each layer stretched by a factor of 1.2 and the total thickness of the five prism layers together was 0.4 mm. This ensured that the temperature gradients at the interfaces were resolved sufficiently and also assisted with the stability of the solution. The prism layers at the interfaces between the fuel compact and the moderator graphite, and the between the moderator graphite and the coolant channel can be seen in Fig. 4. The conformal mapping across the interfaces can also be seen.

A surface growth rate of 1.3 was added with a minimum size of 25 % and maximum size of 100 % of the base size of the mesh. The polyhedral density and growth rates were both set as 1, as well for the volume blending. The mesh for the 1200 mm section of the SCFM could then be created. This mesh was then used as the basis and the mesh required for the simulations was generated by employing extrusion.

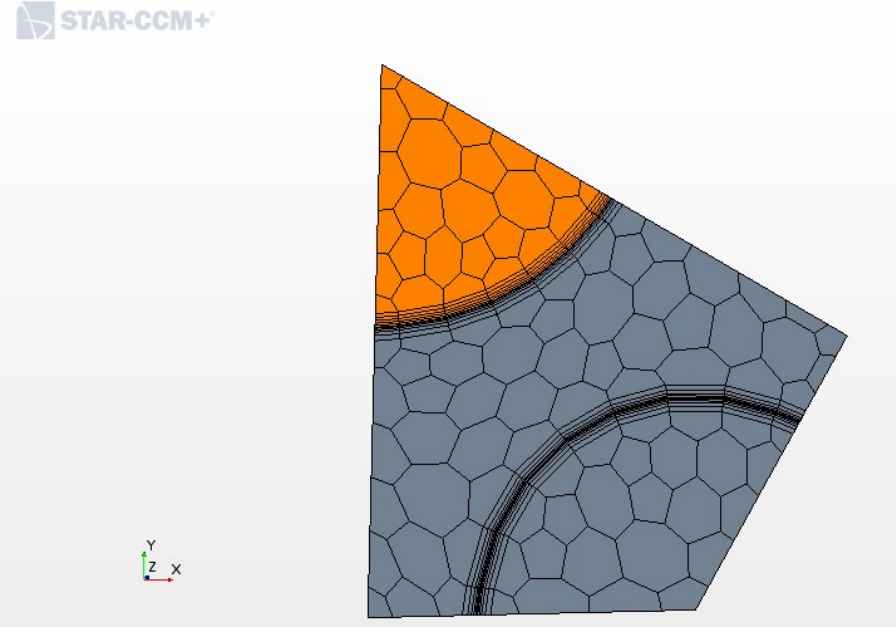

Fig. 4: Cross section of STAR CCM+ (Full 3D) mesh.

The total length of the 1/6th SCFM model required for the simulations is 9200 mm to account for the top reflector and the height of 8000 mm of the heated section as shown in Fig. 1(a). Travis & El-Genk (2013b) commented that the 1200 mm length of the reflector is 75 coolant channel diameters which is sufficient for fully developed turbulent flow to occur at the entrance to the heated section.

The extrusion took place from the top faces of the fuel compact, moderator graphite and the coolant channel regions of the original 1200 mm section of the SCFM. The extrusion was specified to be normal to the surfaces and to consist of 11500 layers. With the length of the extrusion is set as 9200 mm this resulted in each layer having a thickness of 0.8 mm. This was similar to the size of the axial increments used by Travis & El-Genk (2013b). After the extruded mesh was generated the original volume mesh region was deleted from the simulation. New interfaces between the fuel compact and the moderator graphite regions and between the moderator graphite and the coolant channel regions then had to be created. This approach ensured that the mesh distribution in the axial direction was uniform and structured and unstructured in the radial direction. This improved the stability of the solution.

The fuel compact region extended over the entire height of the SCFM model which resulted in the fuel compact properties also being assigned to the section in the reflector at the top of the model. This had a negligible effect because no volumetric heat source was specified in this section. Field functions were defined to specify the appropriate heat sources in the fuel compact region of the heated section to account for the required power profile.

At the top of the coolant channel an inlet temperature of 914 K was specified, whilst an outlet pressure of 7.022 MPa was prescribed at the bottom of the coolant channel. The simulation was initialized by specifying a velocity inlet boundary condition of 41.5114 m/s which corresponded to the specified mass flow rate of 0.0051 kg/s. This helped to stabilize the solution during the initial phase of the simulation. Once the solution has stabilized the velocity boundary condition was replaced with the specified mass flow rate condition. The outside surfaces of the fuel compact and the moderator graphite were specified to be adiabatic and the outside surfaces of the coolant channel as symmetry planes.

The calculated heat flux at the coolant channel wall, the wall temperatures and the bulk fluid temperatures were then used to obtain the local convection heat transfer coefficient. These values were found to in good agreement with the corresponding values predicted by the Travis and EL-Genk correlation. The simulation was also tested on a coarser grid with an axial increment height of 1.6mm. Unlike Travis & El-Genk (2013b) convergence was obtained. However, the values for the local heat transfer coefficient differed by over 10 % from the corresponding values predicted by the Travis & El-Genk correlation (2013b).

1D Flownex Model

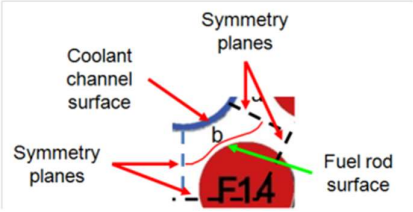

Fig. 5: Unit cell for conduction between fuel compact and coolant channel.

Flownex is a system level simulation tool for the modeling of thermal-fluid systems and is used many industries in the design and optimization of thermal-fluid systems. Flownex calculates amongst others the flow rates, temperatures, pressures and heat transfer rates using both steady state and transient models (M-Tech Industrial, 2017).

In the Flownex model only 1/6th of the SCFM as shown in Fig. 5 was considered due to the symmetry that can be observed in Fig.1 (b). The model therefore consists of 1/6th of a coolant channel, 1/3rd of a fuel compact and the associated graphite matrix or moderator (Nel & Du Toit, 2018). The conduction heat transfer through the fuel compact is modelled using one conduction component as shown in Fig. 6. The conduction heat transfer through the graphite from the surface of the fuel compact to the surface of the coolant channel is modelled using two conduction components. A convection component is used to model the convection heat transfer

between the wall of the coolant channel and the helium in the coolant channel. The convective heat transfer and flow in the coolant channel is modelled using a pipe component. The appropriate conduction lengths and areas for the graphite conduction components were obtained from the shape factors calculated by Sambureni (2015). The material properties given in Table 1 were implemented in the Flownex model using material property tables. The correlations (2) and (3) for the turbulent forced convection Nusselt number were implemented through the appropriate user coding in the convection component.

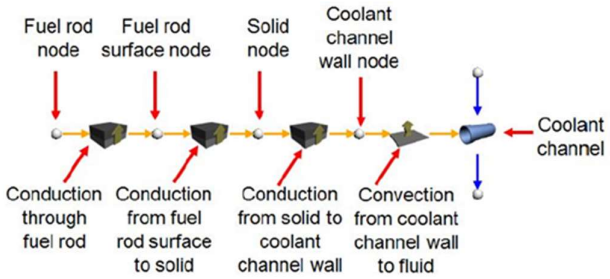

Fig. 6: Flownex network for heat transfer from fuel rod and coolant channel

The SCFM was discretized into 10 primary increments each with a length of Δz = *800 mm* as shown in Fig. 7 with the inlet boundary at the top and the outlet boundary at the bottom. To account for the inlet effects three discretization strategies were tested by Nel and Du Toit (2018). In two cases the first increments (top increment in Fig. 7) was divided into two increments of 400 mm each. The first of the two increments corresponding to *z/D≤25* was then subsequently subdivided into five or ten increments with lengths of 80 mm and 40 mm respectively.

The total power input for the 1/6th SCFM is 9.233 kW_{th}. When a uniform power profile is considered the power input *P(z)* for an increment Δz *is* given as:

$$P(z) = PD_{ma} \ . A . \Delta z \tag{3}$$

with PD_{max}= *27.403* MW_{th}/m^3 and *A* the cross sectional area of the 1/3rd fuel compact.

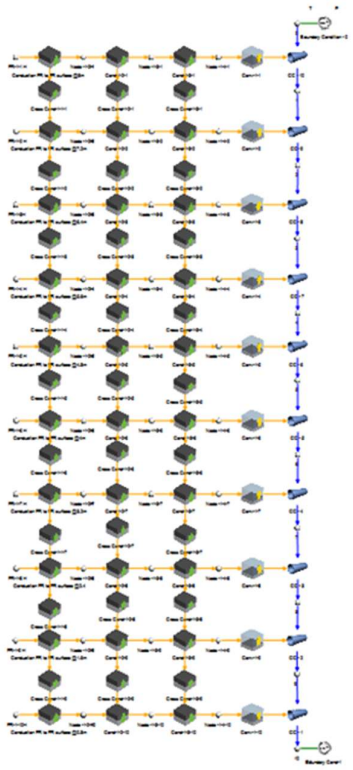

Fig. 7: 800 mm x 10 increment Flownex model

In the case of the cosine power profile the power input for an increment Δz *is* given as:

$$P(z) = PD_{max}.A.\left[2.47373\sin(0.2566z) - 3.01134\cos(0.2566z)\right]_{z-\frac{1}{2}\Delta z}^{z+\frac{1}{2}\Delta z} \qquad (4)$$

where PD_{max}= *33.135* MW_{th}/m^3. The axial position *z* is taken in the middle of the increment.

The power for an increment is distributed to the two fuel nodes associated with the increment according to the volumes of the control volumes represented by the nodes. This leads to 25 % of the power allocated to the first node and 75 % of the power allocated to the surface node.

A pressure of 7.07 MPa and a temperature of 914 K were specified at the inlet and an out flowing mass flow rate of 0.0051 kg/s at the outlet.

Nel & Du Toit (2018) found that local heat transfer coefficients, coolant wall temperatures and the bulk fluid temperatures predicted by the Flownex model to be in very good agreement with values predicted by Travis & El-Genk (2013b).

3D ANSYS Fluent/ 1D Flownex Model

Following Travis & El-Genk (2013b) the SCFM was also modelled using a 3D CFD approach to simulate the heat transfer in the fuel compact and the moderator graphite and a 1D approach to simulate the fluid flow in the coolant channel. Thus the fuel compact and the moderator graphite model were modelled in ANSYS Fluent and the coolant channel was modelled in Flownex. During the simulation the two codes transfer the relevant data between each other.

3D ANSYS Fluent

The coupled 3D ANSYS Fluent / 1D Flownex only considered the 8000 mm heated section of the SCFM due to the fact that Flownex only accounts for the mass flow rate and the associated average velocity in the coolant channel and the reflector block is therefore not needed for the velocity profile to develop. The 3D geometry of the SCFM consisting of the fuel compact and the moderator graphite was assembled in Solidworks and imported into ANSYS Fluent. The model was then divided into twenty axial increments or regions corresponding to the axial increments of the Flownex model described in the next section. The different increments of the fuel compact and the moderator graphite were then categorized and labeled and all the increment faces named to reflect their nature, e.g. symmetry planes and walls.

In the ANSYS Fluent model prism layers were only defined on the moderator side of the coolant channel wall. Five prism layers were specified and the layer thickness of all the layers was chosen such that each layer stretched by a factor of 1 and the total thickness of the combined five prism layers was 0.4 mm. The tetrahedral meshing method was used for the entire model of the SCFM. A single mesh continuum was generated for all the components in the model. A basic base size of 3 mm was chosen for the mesh. To ensure that the cells faces on the interfaces between adjacent parts and sub-assemblies matched to form a conformal mesh throughout the entire model, a sweep and assembly meshing scheme was used. This ensured that no interpolation was necessary for the transfer of information across interfaces. The conformal mapping across the interfaces and prism layers of the generated mesh can be seen in Fig. 8 and also illustrates the mesh that was set up in ANSYS Fluent for the moderator graphite and fuel compacts.

After the mesh was completed all the different boundary conditions and input and output parameters were set and assigned. The material properties that were defined were taken from Table 1. The thermal conductivity correlations for the different materials, however, had to be rewritten as polynomials since ANSYS Fluent only uses polynomials as input equations for the material properties. The power profiles were applied as heat flux sources on the two symmetry planes of the fuel compacts. The volumetric heat source data was therefore converted to the

heat flux data to give the exact amount of heat that was needed for each increment of the fuel compact.

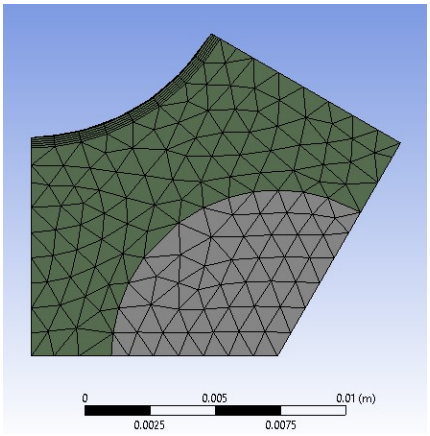

Fig. 8: Cross section thought ANSYS Fluent mesh.

Flownex

With the flow in the coolant channel being simulated in Flownex the same number and size of increments were employed as those that were used in the finest Flownex mesh by Nel & Du Toit (2018). The first 10 increments is each 40 mm in length followed by one 400 mm increment and then nine increments each 800 mm in length. This model uses only the convection and pipe elements and their corresponding nodal elements to represent the flow and convection heat transfer in the channel. The Nusselt number correlation proposed by Travis & El-Genk (2013b) was again used to obtain the local heat transfer coefficients. The properties of the convection and pipe components were the same as those that were used in full 1D Flownex simulation (Nel & Du Toit, 2018).

The boundary conditions at the inlet and the outlet of the coolant channel were the same as the corresponding boundary conditions used by Nel & Du Toit (2018) for the full 1D Flownex simulation. The inlet and the first increment of the Flownex model are shown in Fig. 9.

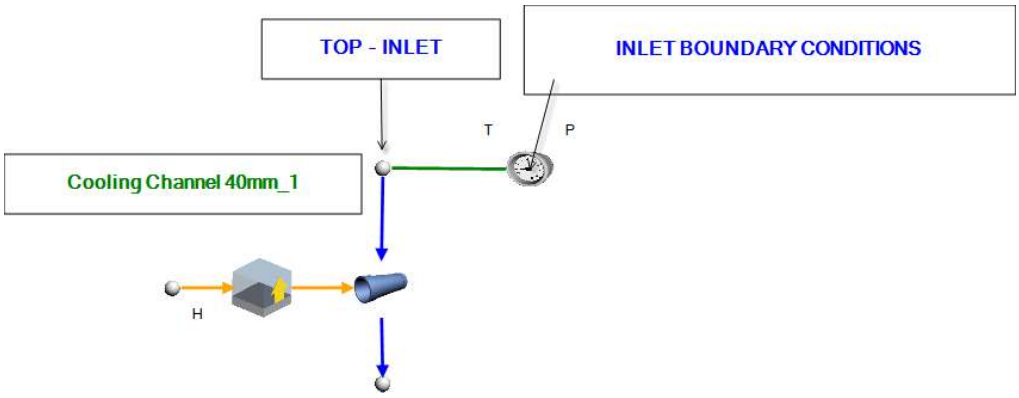

Fig. 9: Inlet and first increment of the coupled Flownex model.

Coupled ANSYS Fluent/Flownex

After the 3D CFD model of the fuel compacts and the moderator graphite and the 1D SCFD model of the coolant channel were constructed the two models had to be coupled to each other in order to define the data to be transferred between the two models. From the coupling strategies that are available it was chosen to select Flownex as the managing or controlling code and to run ANSYS Fluent in server mode.

In Flownex a Server Fluent Generic Interface component was inserted to control the transfer of data between Flownex and ANSYS Fluent. In the interface component the ANSYS Fluent case file that was used was selected as well as the location of the aas key file that is needed for Flownex to be able to access ANSYS Fluent. In the coupling between the codes the heat released by the various coolant channel wall increments in the ANSYS Fluent model is transferred to the relevant Flownex coolant channel wall nodes, and the coolant channel wall nodes temperatures obtained by Flownex are transferred to the relevant ANSYS Fluent coolant channel wall increments.

In the Flownex model data transfer links that couple the Flownex wall nodes to the relevant ANSYS Fluent input / output parameters to account for the exchange of the coolant channel wall temperatures and the heat released by the coolant channel wall are defined. In Fig. 10 a representation can be seen of the main coupled model in Flownex where the dashed lines signify the data transfer links to and from ANSYS Fluent represented by the yellow and black ANSYS component. Data transfer links must be defined for all the increments.

Before the coupled model was run both the 3D ANSYS Fluent and the 1D Flownex models were executed separately using fixed boundary conditions to create initial solutions. The coupled simulation could then be started. In Flownex the wall temperature is transferred to ANSYS

Fluent to the corresponding increments. In ANSYS Fluent a heat transfer report is used to transfer information for the heat generated to the corresponding elements in Flownex.

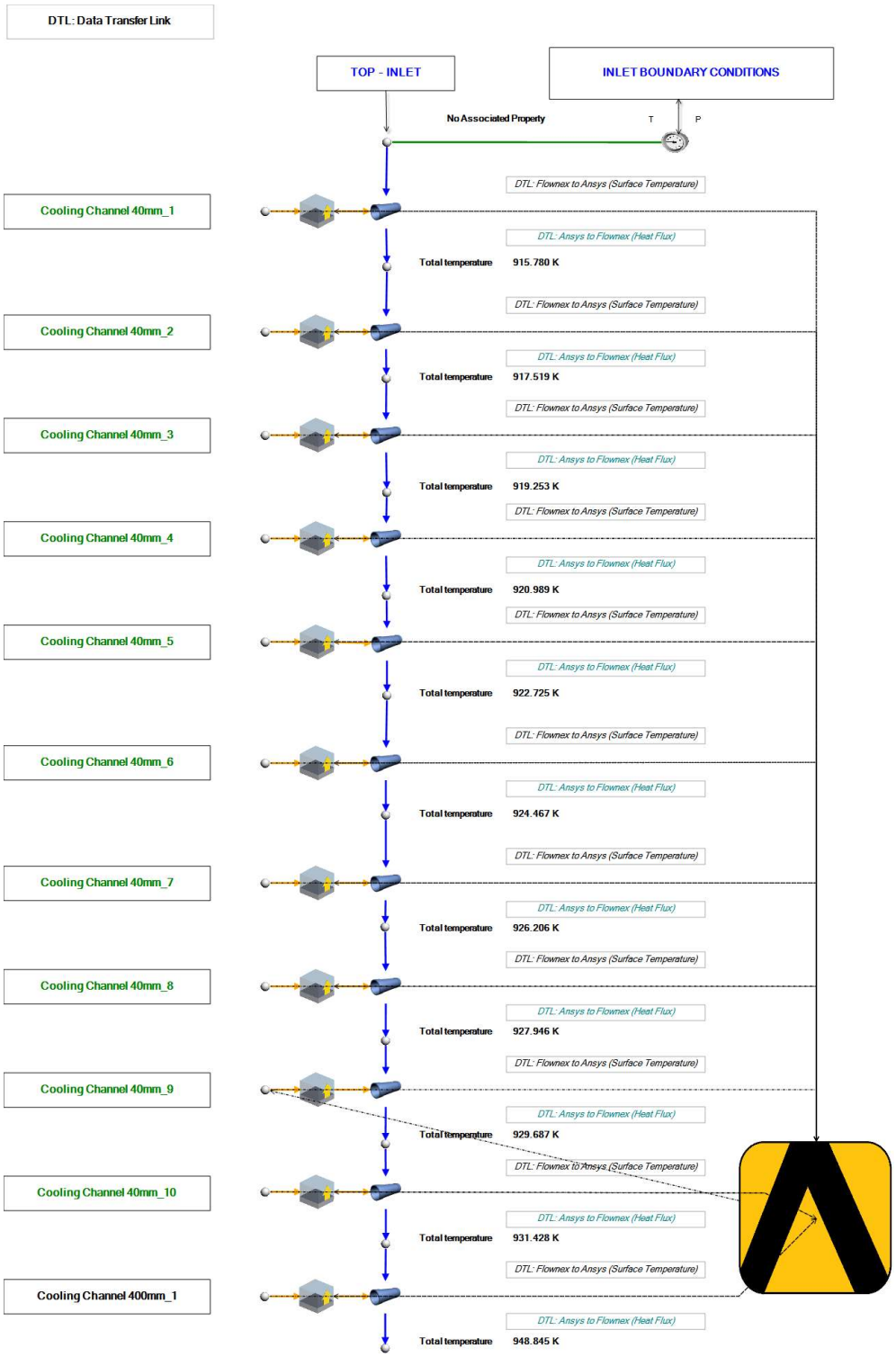

Fig. 10: FNX/ANSYS Fluent coupling.

After the coupling of the two codes an iterative procedure was followed in which Flownex completed a number of iterations and then transfers the necessary data to ANSYS Fluent. ANSYS Fluent then initialized with the inputs received and completed a number of iterations before the necessary information was sent back to Flownex. This procedure was repeated until the specified convergence criteria were met.

Results

The results obtained by the full 3D CFD STAR CCM+ model are first compared with the corresponding full 1D Flownex results (Nel & Du Toit, 2018) which are in very good agreement with the reconstructed Travis & El-Genk (2013b) results. All the results only pertain to the heated section of the SCFM and z = 0 mm is at the entrance to the heat section.

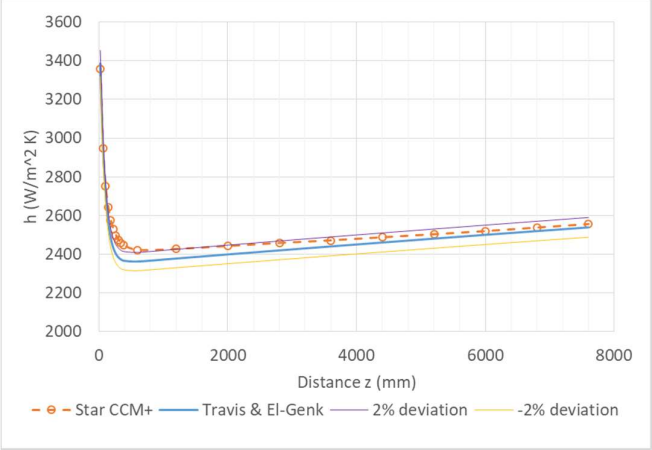

Fig. 11: Local heat transfer coefficients as a function of axial position for the uniform power profile.

In Fig. 11 the local heat transfer coefficient as a function of axial position obtained by the full 3D model for the uniform power profile is compared with the corresponding variation obtained by the full 1D model. The agreement is good and over 90 % of the length within the fitting accuracy of 2 % associated with the heat transfer correlation proposed by Travis & El-Genk (2013b).

The variation of the local heat transfer coefficient as a function of axial position obtained by the full 3D model for the cosine power profile is compared with the corresponding variation obtained by the full 1D model in Fig. 12. The agreement is again good and over 90 % of the length with the fitting accuracy of Travis & El-Genk correlation. The results also exhibit the same trend as the Travis & El-Genk (2013b) results shown in Fig. 3. In both Fig. 11 and 12 the maximum deviation between the corresponding results occurs in the transition between the inlet region and the fully developed region.

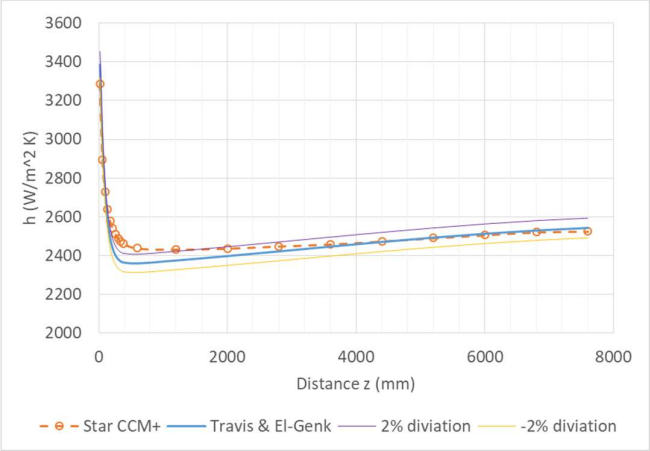

Fig. 12: Local heat transfer coefficients as a function of axial position for the cosine power profile.

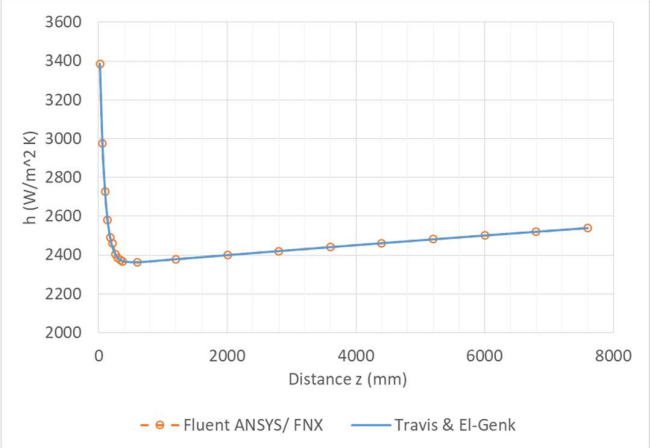

Fig. 13: Local heat transfer coefficients as a function of axial position for the uniform power profile.

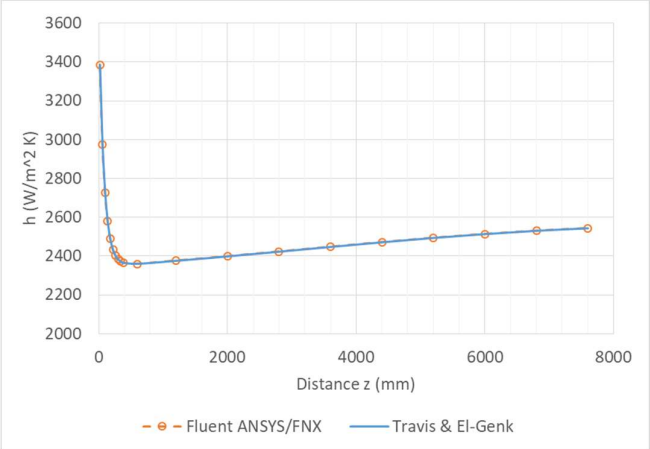

Fig. 14: Local heat transfer coefficients as a function of axial position for the Cosine power profile.

Next the results obtained by the coupled 3D ANSYS Fluent /1D Flownex model are compared with the corresponding full 1D Flownex results (Nel & Du Toit, 2018).

In Fig. 13 the local heat transfer coefficient as a function of axial position obtained by the coupled 3D/1D model for the uniform power profile is compared with the corresponding variation obtained by the full 1D model. The agreement is very good. The variation of the local heat transfer coefficient as a function of axial position obtained by the coupled 3D/1D model for the cosine power profile is compared with the corresponding variation obtained by the full 1D model in Fig. 14. The agreement is again very good. The same deviations observed in Fig. 11 and Fig. 12 will therefore occur between the results for the current full 3D model and the corresponding results of the coupled 3D/1D model.

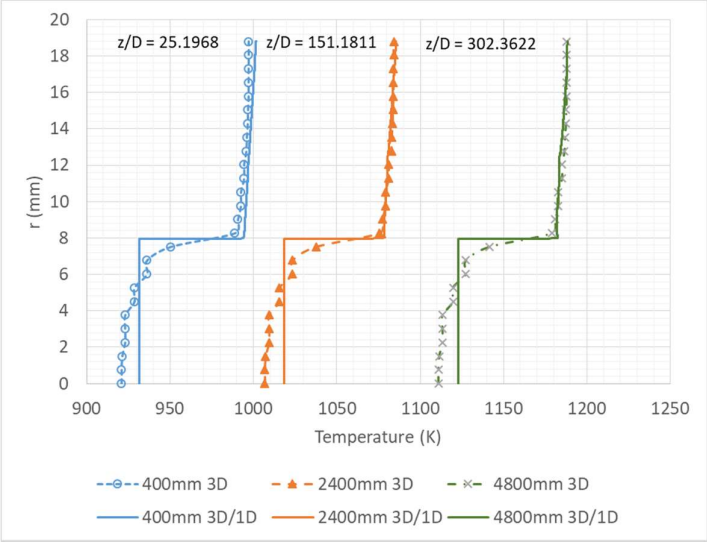

Fig. 15: Radial temperature distribution at three axial locations for the 3D and 3D/1D Uniform power profile

Fig. 15 and Fig. 16 show the radial temperature distributions at three different axial locations within the SCFM obtained by the full 3D model and by the coupled 3D/1D model. The radial distributions are from the centre of the coolant channel to the centre of the fuel compact. The results shown in Fig. 15 are for a uniform power profile, whilst the results shown in Fig. 16 are for a cosine power profile.

It can be seen in both Fig. 15 and Fig. 16 that the agreement between the temperature variations for the fuel compact and the moderator graphite predicted by the full 3D model and the coupled 3D/1D model is very good. The moderator graphite and fuel compact occurs between 7.94 mm and 18.8 mm on the graphs. A comparison between the temperature variations for the coolant channel predicted by the full 3D model and the coupled 3D/1D model reflect the fact that the coupled 3D/1D model can only predict the bulk fluid temperature.

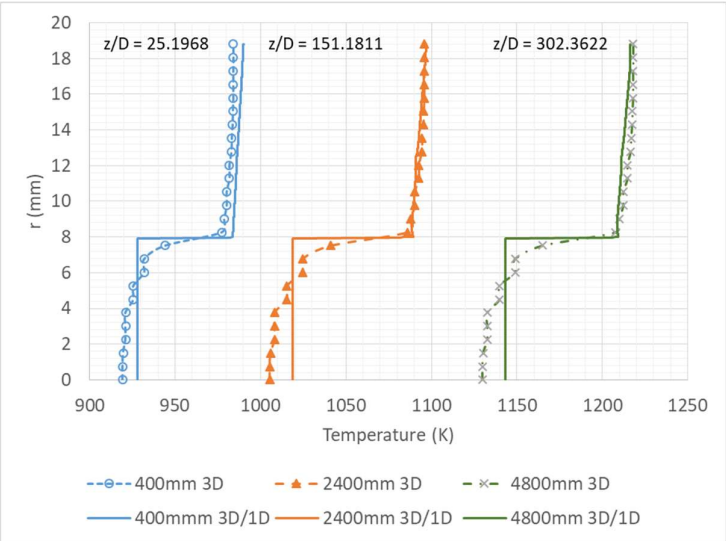

Fig. 16: Radial temperature distribution at three axial locations for the 3D and 3D/1D Cosine power profile

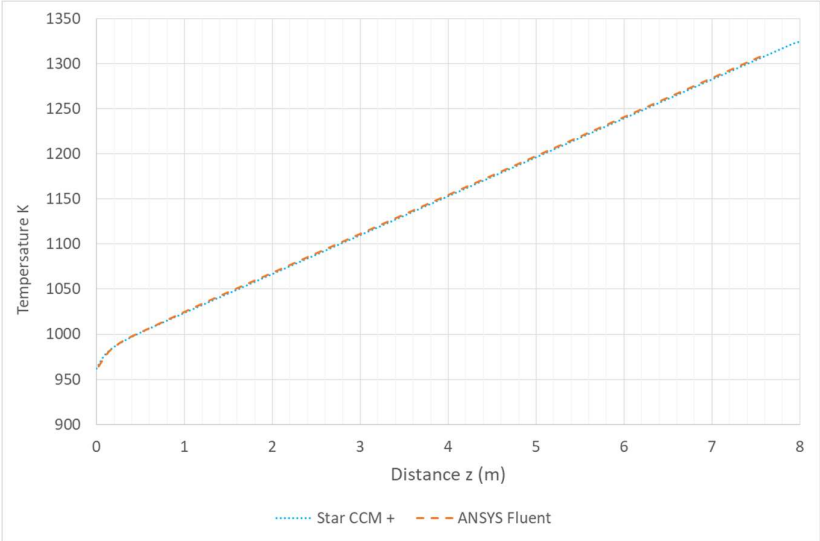

Fig. 17: Fuel compact temperature as function of axial position for the uniform power profile.

In Fig. 17 the axial variation in the temperature at the center of fuel compact for the uniform power profile as obtained in the full 3D model is compared with the corresponding results obtained by the 3D/1D model. It can be seen that the agreement is very good. The axial variation in the temperature at the center of the fuel compact for the cosine power profile as obtained from the full 3D simulation is compared with the corresponding temperature profile obtained by the 3D/1D simulation in Fig. 18. Again the agreement is very good.

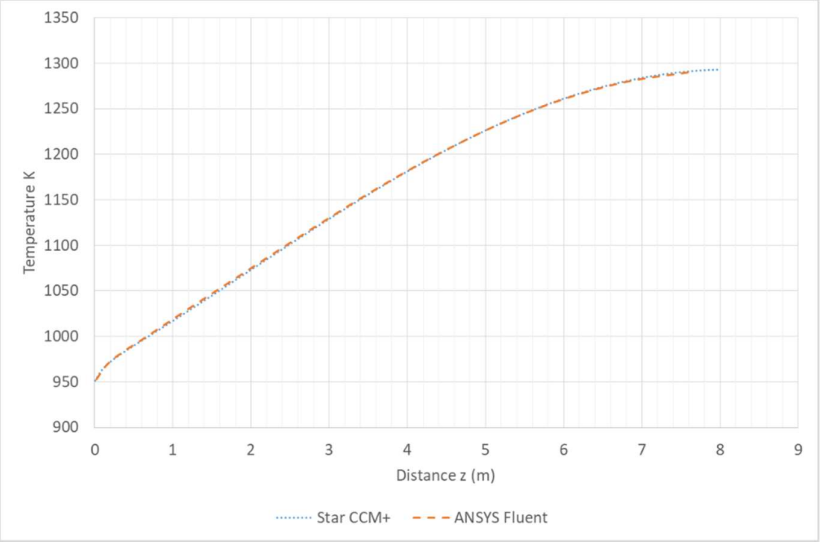

Fig. 18: Fuel compact temperature as function of axial position for the cosine power profile.

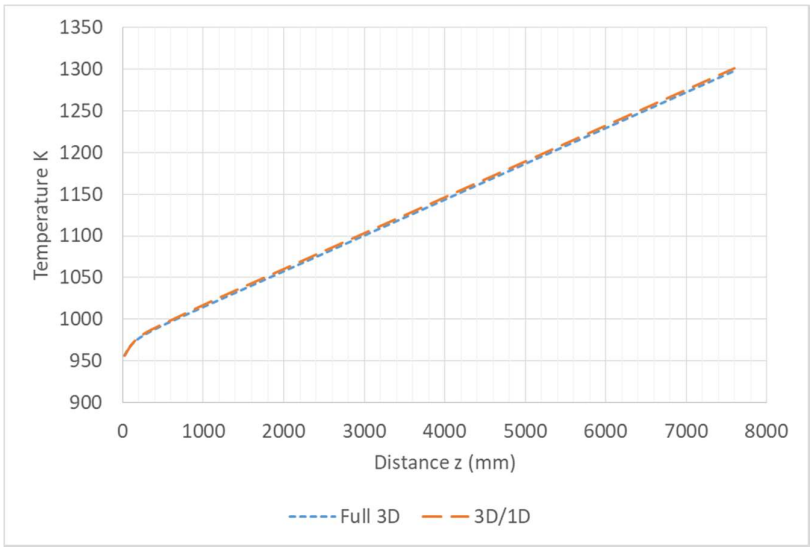

Fig. 19: Coolant channel wall temperature as function of axial position for the uniform power profile.

The coolant wall temperatures provided by each Flownex increment in the coupled 3D/1D simulation to the corresponding ANSYS Fluent coolant wall increments are received by ANSYS Fluent as a constant value for each wall increment. This then results in a stepwise constant profile for the axial variation in the boundary values employed by ANSYS Fluent for the coolant wall temperature. This is also reflected in the values obtained by the 3D/1D model for the variation in the temperatures along the centerline of the fuel compact. The temperature profiles shown in Fig. 17 and Fig. 18 are based on the temperatures that were extracted at the centre of each axial increment.

The comparison between the results obtained by the full 3D model and the 3D/1D model for the variation in the axial direction of the coolant wall temperature for the in uniform power profile is shown in Fig. 19. It can be seen that the agreement between the corresponding results is very good.

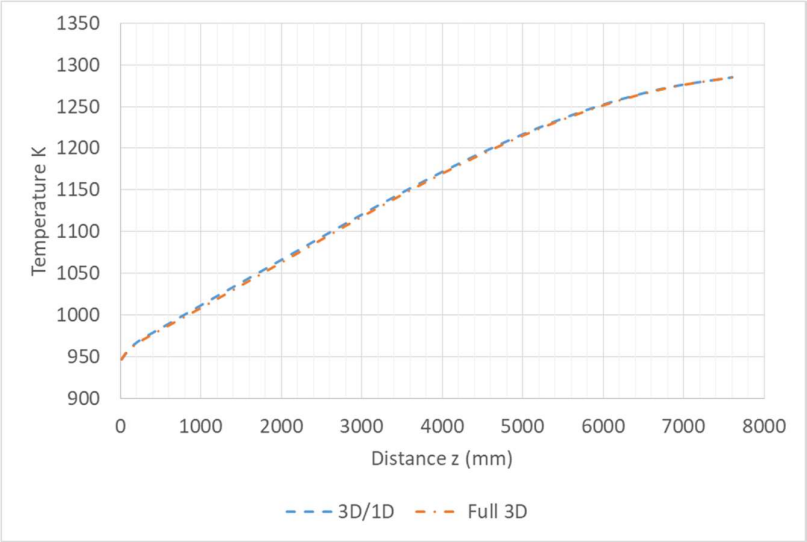

Fig. 20: Coolant channel wall temperature as function of axial position for the cosine power profile.

Fig. 20 shows the comparison between the results obtained by the 3D model and the 3D/1D model for the variation in the axial direction of the coolant wall temperature for the cosine power profile. It can again be seen that the agreement between the corresponding results is very good. The effect on the temperatures of the development length at the entrance of the heated section can be seen in the shape of the temperature profiles at $0 < z < 400$ mm in both Fig. 19 and Fig. 20.

Conclusions

The conduction heat transfer through the prismatic blocks containing the fuel elements in a Very High Temperature Reactor (VHTR) is of crucial importance for the proper operation of the reactor under normal operating conditions and upset conditions. This paper discussed full 3D, full 1D and 3D/1D coupled CFD models that were used to simulate the heat transfer and fluid flow in a SCFM of a prismatic fuel block.

The study performed by Travis & El-Genk (2013b) on the numerical modeling of the heat transfer and flow in a SCFM was used as the basis for the current study. The study also

implemented the full 1D model of 1/6th of the SCFM developed by Nel & Du Toit (2018). The purpose of the study was to evaluate the coupling of the CFD code ANSYS Fluent and the network or SCFD code Flownex to perform a coupled 3D/1D simulation of the SCFM.

The models developed in this study considered only 1/6th of the cross section of the SCFM simulated by Travis and El-Genk (2013b) due to the symmetry in the geometry of the SCFM. An explicit 3D model of the fuel compact, graphite moderator and coolant channel of the 1/6th of the SCFM was created using the CFD code STAR CCM+. The calculated heat flux at the coolant channel wall, the wall temperatures and the bulk fluid temperatures were then used to obtain the local convection heat transfer coefficient. These values were found to be in good agreement with the corresponding values predicted by the Travis and El-Genk correlation.

The coupled 3D/1D model of the 1/6th SCFM was generated using ANSYS Fluent for the explicit 3D representation of the fuel compact and graphite moderator and the system CFD Flownex for the 1D representation of the coolant channel. The Travis and El-Genk convection heat transfer correlation was employed to couple the heat transfer between the 3D and 1D formulations. The results for the coolant channel wall temperatures and fuel compact centre temperatures were found to be in very good agreement with corresponding values predicted by the full 3D explicit model.

It can therefore be concluded that the coupled 3D/1D simulation provide results for the coolant wall and fuel compact temperatures that are in very good agreement with the coolant wall and fuel compact temperatures obtained by the full 3D simulation. Due to the marked reduction in the grid requirements for the modelling of the flow in the coolant channel in the coupled 3D/1D model compared to the full 3D model, it also leads to reduction in the computational resources required.

However, the coupling strategy used to transfer the relevant data between ANSYS Fluent and Flownex should be investigated to determine whether it might be possible to transfer a smoother profile for the coolant wall temperatures to be used as boundary conditions in ANSYS Fluent. More increments can be implemented in the 1D Flownex model, but the methodology to define the coupling should receive attention to determine to what extent it can be automated.

Acknowledgements

This work is based on the research supported by the South African Research Chairs Initiative of the Department of Science and Technology and the National Research Foundation of South Africa (Grant No 61059).

Any opinion, findings and conclusion or recommendation expressed in this material is that of the authors and the NRF does not accept any liability in this regard.

The first author would like to thank Mr OC Koekemoer for his invaluable assistance.

References

B. W. Travis and M. S. El-Genk, “Thermal–hydraulics analyses for 1/6 prismatic VHTR core and fuel element with and without bypass flow,” *Energy Conversion and Management,* vol. 67, p. 325–341, 2013a.

GJ Nel, C. d. “T. A System CFD model of a Single-Channel Fuel Module in a Prismatic Block VHTR”. SACAM, vol. 11, 2018.

J. H. Lee, H. K. Cho and G. C. Park, “Development of the loss coefficient correlation for cross flow between graphite fuel blocks in the core of prismatic very high temperature reactor-PMR200,” *Nuclear Engineering and Design,* vol. 307, p. 106–118, 2016.

M. H. Kim and H. S. Lim, “Evaluation of the influence of bypass flow gap distribution on the core hot spot in a prismatic VHTR core,” *Nuclear Engineering and Design,* vol. 241, p. 3076–3085, 2011.

M-Tech Industrial, Flownex SE Version 8.8.1.3243, http://www.flownex.com, 2017

N. I. Tak, M. H. Kim and W. J. Lee, “Numerical investigation of a heat transfer within the prismatic fuel assembly of a very high temperature reactor,” *Annals of Nuclear Energy,* vol. 35, p. 1892–1899, 2008.

N. I. Tak, M. H. Kim, H. S. Lim and J. M. Noh, “A practical method for whole-core thermal analysis of a prismatic gas-cooled reactor,” *Nuclear Technology,* vol. 177, pp. 352-365, 2012.

N. I. Tak, S. N. Lee, M. H. Kim, H. S. Lim and J. M. Noh, “Development of a core thermo-fluid analysis code for prismatic gas cooled reactors,” *Nuclear Engineering and Technology,* vol. 46, no. 5, pp. 641-654, 2014.

O. Cioni, M. Marchand, G. Geffraye and F. Ducros, “3D thermal-hydraulic calculations of a modular block-type HTR core,” *Nuclear Engineering and Design,* vol. 236, p. 565–573, 2006.

P. G. Rousseau, P, G. P. Greyvenstein, (2002). Modelling of the HTTR in Flownex. *Proceedings of the 1st International Topical Meeting on High Temperature Reactor Technology*, Petten, Netherlands.

R. W. Johnson and H. Sato, “Bypass flow computations using a one-twelfth symmetric sector for normal operation in a 350 MWth prismatic VHTR,” *Nuclear Engineering and Design,* vol. 251, p. 84–91, 2012.

P. Sambureni, (2015). *Thermal fluid network model for a prismatic block in a gas-cooled reactor using FLOWNEX*, Masters dissertation, North-West University.

SIEMENS, STAR CCM+. https://mdx.plm.automation.siemens.com/star-ccm-plus, 2018

S. Maruyama, K. Yamashita, N. Fujimoto, I. Murata and Y. Sudo, “Evaluation of Hot Spot Factors for Thermal and Hydraulic Design of HTTR,” *Journal of Nuclear Science and Technology,* vol. 30, no. 11, pp. 1186-1194, 1993.

S. Maruyama, N. Fujimoto, Y. Sudo, T. Murakami and S. Fujii, “Evaluation of core thermal and hydraulic characteristics of HTTR,” *Nuclear Engineering and Design,* vol. 152, pp. 183-196, 1994.

S. N. Lee, N. I. Tak, M. H. Kim and J. M. Noh, “Thermal analysis of prismatic gas-cooled reactor core under coolant channel blockage accidents,” *Annals of Nuclear Energy,* vol. 71, p. 11–22, 2014.

S. J. Yoon, J. H. Lee, M. H. Kim and G. C. Park, “The effects of cross flow gap and axial bypass gap distribution on the flow characteristics in prismatic VHTR core,” *Nuclear Engineering and Design,* vol. 250, p. 465–479, 2012.

Travis, BW, El-Genk, MS “Numerical Simulation and Turbulent Convection Heat Transfer Correlation for Coolant Channels in a Very-High-Temperature Reactor”. *Heat Transfer Engineering*, 34, no 1, pp. 1–14. (2013b).

Y. H. Tung, Y. M. Ferng, R. W. Johnson and C. C. Chieng, “Study of natural circulation in a VHTR after a LOFA using different turbulence models,” *Nuclear Engineering and Design,* vol. 263, p. 206– 217, 2013.

Y. H. Tung, Y. M. Ferng, R. W. Johnson and C. C. Chieng, “Transient LOFA computations for a VHTR using one-twelfth core flow models,” *Nuclear Engineering and Design,* vol. 301, p. 89–100, 2016.

CHAPTER 4: CONCLUSIONS AND RECOMMENDATIONS

The accurate numerical simulation of the conduction and convection heat transfer and fluid flow in the prismatic fuel blocks of a Very High Temperature Reactor is of crucial importance for the design and evaluation of the performance of the reactor under normal and upset conditions. The study performed by Travis & El-Genk (2013b) on the numerical modeling of the heat transfer and flow in a SCFM was used as the basis for the current study.

The models developed in this study considered only 1/6th of the cross section of the SCFM simulated by Travis & El-Genk (2013b) due to the symmetry in the geometry of the SCFM. Full 1D, full 3D and 3D/1D coupled CFD models were developed to simulate the heat transfer and fluid flow in the 1/6th of the SCFM.

Nel & Du Toit (2018) developed a full 1D model of 1/6th of the SCFM employing the system CFD code Flownex. The Nusselt number correlation derived by Travis & El-Genk (2013b) was implemented in the model and uniform and cosine power profiles were considered. Variations in the coolant channel wall temperature, bulk fluid temperature and the local heat transfer coefficient were extracted from the simulations. With the different sets of results obtained it was found these results were in very good agreement with the corresponding results obtained by Travis & El-Genk (2013b). It was concluded that the Nusselt number correlation was implemented successfully into Flownex and that the heat transfer coefficient was predicted with a very good accuracy. Based on the accuracy of the predicted results it is concluded that the 1D Flownex model is capable of predicting the wall and fluid temperatures occurring in the SCFM accurately.

The CFD code STAR CCM+ was used to create an explicit 3D model of the fuel compact, graphite moderator and coolant channel of the 1/6th of the SCFM. The local convection heat transfer coefficient was calculated by using the heat flux at the coolant channel wall, the wall temperatures and the bulk fluid temperatures predicted by the simulations. These values were found to be in good agreement with the corresponding values predicted by the Travis and El-Genk correlation. It was found that for the local heat transfer coefficient for the uniform and cosine power profile the results are within 2 % of the predicted values over 90 % of the length of the SCFM.

The full 3D and the full 1D models were combined by developing a coupled 3D/1D model. The 3D CFD code ANSYS Fluent was employed to simulate the heat transfer in the fuel compact and the moderator graphite and the fluid flow in the coolant channel was simulated using the1D system CFD code Flownex. The Travis & El-Genk convection heat transfer correlation was

employed to couple the heat transfer between the 3D and 1D formulations. ANSYS Fluent provided the heat flux at the coolant channel wall to Flownex, whilst Flownex provided the temperature of the coolant channel wall to ANSYS Fluent.

The results for the coolant channel wall temperatures and fuel compact centre temperatures were found to be in very good agreement with the corresponding values predicted by the full 3D explicit model. Results were extracted from three different locations for the radial temperature from the centre of the coolant channel to the centre of the fuel compact. These results from the 3D/1D model were compared with the full 3D model results and it was found that the results are in very good agreement. It was also found that the axial temperature variations predicted by the 3D and 3D/1D models of the fuel compact for both uniform and the cosine power profiles were in good agreement. It can thus be concluded that the results obtained with the 3D/1D model are valid and accurate.

It was noted that 3D/1D coupling resulted in a marked reduction in the grid requirements for the modeling of the coolant channel compared to the full 3D model. This also led to a reduction on the computational resources required. Thus the coupling of the two codes, to make a 3D/1D model of the $1/6^{th}$ of the SCFM, can be considered successful.

It is recommended that the coupling strategy employed to exchange the relevant information between ANSYS Fluent and Flownex should be investigated to determine whether a smoother profile for the coolant wall temperatures to be used as boundary conditions in ANSYS Fluent could be transferred. Although more increments could be used in the 1D Flownex model, the methodology to define the coupling should receive attention to determine to what extent it can automated.