Computational Study of Convective Heat Transfer in Porous Media by Finite Volume Method

[1]Ranjan Kumar Mishra, [2]Dr. Bharat B Gupta, [3]Dr.Prabal Talukdar

[1]M.Tech Scholar, [2] Distinguished Professor & HOD, [3]Associate Professor

[1&2]Department of Mechanical Engineering,
[1&2]Sharda University, Greater Noida, India
[3]Indian Institute of Technology-DELHI

***Abstract*—A computational investigation has been carried out for accurate determination of heat transfer in metal foams in forced convective flows. In this, using c program codes, we are finding the value of temperature profile data at various nodes using finite volume method. The single-blow method was used, in which, the fluid temperature varies with distance and time and convective heat transfer becomes time-dependent. The method results in a transient and conjugate heat transfer phenomena between the foam and fluid. The convective temperature data thus obtained from generated C program codes are then compared with the theoretical results to obtain the corresponding volumetric heat transfer coefficients between the fluid and foam solid surface. The volumetric coefficients thus determined are compared against available sets of experimental data, so as to examine the consistency among the reported experimental data**.

Keywords- porous media, two energy method, single blow method, c program code

I. INTRODUCTION

A porous medium is a solid structure with interconnected voids. It is the region in space comprising at least two homogeneous material constituents, presenting identifiable interfaces between them in a resolution level, with at least one of the constituents remaining fixed or slightly deformable. In order to use porous media for convective heat exchanger applications, one must estimate the volumetric heat transfer taking place between the solid and fluid phases. Naturally, the local thermal equilibrium assumption must be discarded, and the two energy equation model must be used to determine the distinct temperatures of the fluid phase and solid phase, and the local volumetric heat transfer between the two phases. Thus, for engineering applications of porous media, the evaluation of the volumetric heat transfer coefficients is equally important as that of stagnant thermal conductivities.

II. LITERATURE REVIEW

Fu *et al.* [1] carried out exhaustive measurements and correlated the interstitial heat transfer coefficients of cellular materials, which indicates the linear dependency of the Reynolds number on the Nusselt number. A general correlation for the volumetric heat transfer coefficient was introduced by Kamiuto and Yee [2] for open cellular porous materials, who indicated the Reynolds number exponent for the Nusselt number being close to 0.8. They assembled experimental data reported by Younis and Viskanta [3], Ichimiya [4] and themselves, to find that all these experimental data for the volumetric heat transfer coefficient can be correlated fairly well using the strut diameter as a characteristic length for the open cellular materials. In recent years, some numerical attempts have been made to simulate the heat transport processes in foam materials (Zhiyong Wu [5], A. Kopanidis [6]). Kuwahara and Fumoto [7] conducted a series of three dimensional numerical experiments by assuming a macroscopically uniform flow through the metal foams, so as to estimate the interstitial heat transfer coefficient between the fluid and foam solid surface. Interstitial heat transfer coefficients are usually measured by the "single-blow method" (Liang and Yang [8]). The name “single-blow” is designated because a single fluid is employed in this method to heat the heat transfer surface. As illustrated in Fig. (**1**), in this method, inlet fluid temperature varies with time and convective heat transfer becomes time dependent, which results in a transient and conjugate heat transfer phenomena between the foam and fluid. A fluid flows steadily through a test core. Initially the fluid and the test core have the same uniform temperature. Then a fluid temperature variation is introduced at the inlet. Thereafter, the fluid and solid temperature histories at both inlet and outlet of the test core are measured continuously. These data are then compared with the theoretical results to obtain the corresponding interstitial heat transfer coefficients between the foam of solid surface and the fluid. Details of the method can be found in Wakao and Kaguei [9].

III. SINGLE BLOW METHOD

By using the method of single blow method, we have got the following analysis. In this, an air which is supplied by the blower is constantly heated by the heater. Then the hot air is mixed up well as it passes through a mixer and filter. Thus, the well-mixed hot air flows through the test specimen. And then the further process is carried out. The thermally steady state can be reached after supplying the heated air for 5 to 6 hours.

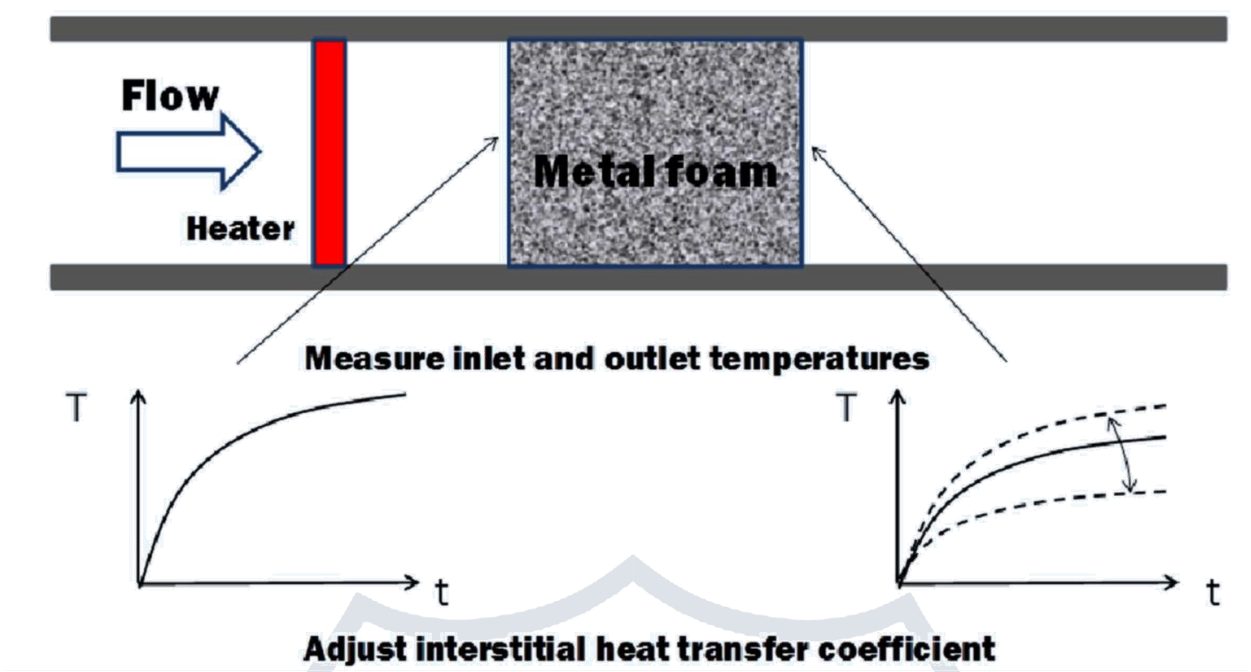
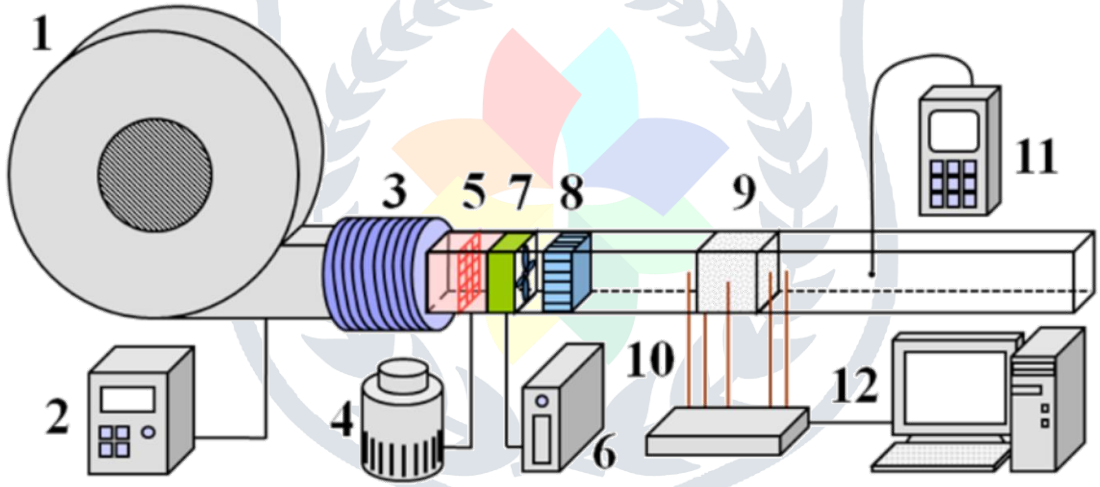


Figure 1: single blow method

Experimental Set-up:
The experimental set-up used in this study is schematically shown in Fig. (**2**).



1 Blower, Blower, 2. Inverter, 3. Connecting duct, 4. Transformer, 5. Heater, 6. AD converter, 7. Mixer, 8. Filter, 9. Test specimen, Thermo-couples, 11. Anemometer, 12. PC

Figure 2: Experimental set-up.

In the current existed experimental set-up, the heat supply is suddenly cut to introduce a fluid temperature variation at the inlet and can be seen. Then, the air and foam temperature variations at both windward and leeward sides are monitored as indicated in the figure according to the given setup. As prescribing the air and foam temperature variations at the inlet of the test specimen from the measurements, both air temperature $\langle T\rangle^f$ and foam temperature $\langle T\rangle^s$ at the leeward side are predicted and correlated with the temperatures predicted using the two-energy equation model:

$$\rho_f c_{pf}\left(\frac{\partial\langle T\rangle^f}{\partial t}+u_D\frac{d\langle T\rangle^f}{dx}\right)=\varepsilon^* k_f\frac{d^2\langle T\rangle^f}{dx^2}-h_v\left(\langle T\rangle^f-\langle T\rangle^s\right) \quad (1)$$

$$\rho_s c_s\frac{\partial\langle T\rangle^s}{\partial t}=(1-\varepsilon^*)k_s\frac{d^2\langle T\rangle^s}{dx^2}-h_v\left(\langle T\rangle^s-\langle T\rangle^f\right) \quad (2)$$

Where u_D and ε^* are the Darcian velocity and effective porosity respectively. The interstitial heat transfer coefficient h_v is adjusted to fit into the measured temperature developments at the leeward side of the test specimen with the predicted temperature responses.

IV. WORK DONE

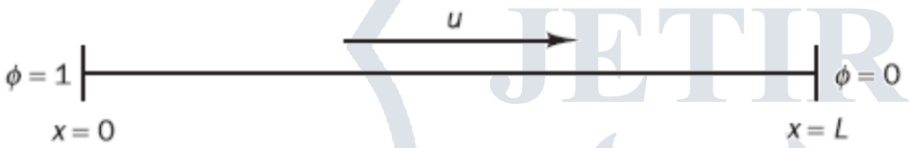
I have developed so many C program codes like one dimensional steady state heat conduction problem from computational fluid dynamics using finite volume method.For example: To solve a one-dimensional convection–diffusion problem we write discretised equations for all grid nodes. This yields a set of algebraic equations that is solved to obtain the distribution of the transported propertyϕ. The process is now illustrated by means of a worked example.

Example 1:

A propertyϕ□is transported by means of convection and diffusion through the one-dimensional domain sketched in Figure 5.2. The governing equation is given below; the boundary conditions are ϕ0 □□1 at *x* □□0 and ϕ*L* □□0 at *x* □□*L*. Using five equally spaced cells and the central differencing scheme for convection and diffusion, calculate the distribution of ϕ□□as a function of *x* for (i) Case 1: *u* □□0.1 m/s, and compare the results with the analytical solution

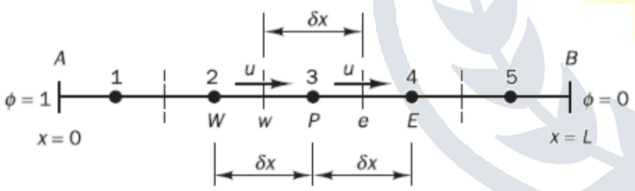
$$\frac{\phi-\phi_0}{\phi_L-\phi_0}=\frac{\exp(\rho u x/\Gamma)-1}{\exp(\rho u L/\Gamma)-1}$$ Equation. (1)

The following data apply: length *L* □□1.0 m, ρ□□□1.0 kg/m3, Γ□□□0.1 kg/m.s.



Solution:

The method of solution is demonstrated using the simple grid shown in Figure 5.3. The domain has been divided into five control volumes giving $\delta x = 0.2$ m. Note that $F = \rho u$, $D = \Gamma/\delta x$, $F_e = F_w = F$ and $D_e = D_w = D$ everywhere. The boundaries are denoted by *A* and *B*.



The discretisation equation and its coefficients apply at internal nodal points 2, 3 and 4, but control volumes 1 and 5 need special treatment since they are adjacent to the domain boundaries.We integrate governing equation and use central differencing for both the diffusion terms and the convective flux through the east face of cell 1.The value ofϕis given at the west face of this cell (ϕ*w* □□ϕ*A* □□1) so we do not need to make any approximations in the convective flux term at this boundary. This yields the following equation for node 1:

$$\frac{F_e}{2}(\phi_P+\phi_E)-F_A\phi_A=D_e(\phi_E-\phi_P)-D_A(\phi_P-\phi_A)$$ Equation. (2)

For control volume 5, the ϕ-value at the east face is known (ϕ*e* □□ϕ*B* □□0). We obtain

$$F_B\phi_B-\frac{F_w}{2}(\phi_P+\phi_W)=D_B(\phi_B-\phi_P)-D_w(\phi_P-\phi_W)$$ Equation. (3)

Rearrangement of equations (5.16) and (5.17), noting that *DA* □□*DB* □□$2\Gamma/\delta x$□□*2D* and *FA* □□*FB* □□*F*, gives discretised equations at boundary nodes of thefollowing form:

with central coefficient

$$a_P = a_W + a_E + (F_e - F_w) - S_P$$

and

$$a_P\phi_P = a_W\phi_W + a_E\phi_E + S_u$$

Node	a_W	a_E	S_P	S_u
1	0	$D - F/2$	$-(2D + F)$	$(2D + F)\phi_A$
2, 3, 4	$D + F/2$	$D - F/2$	0	0
5	$D + F/2$	0	$-(2D - F)$	$(2D - F)\phi_B$

To introduce the boundary conditions we have suppressed the link to the boundary side and entered the boundary flux through the source terms.

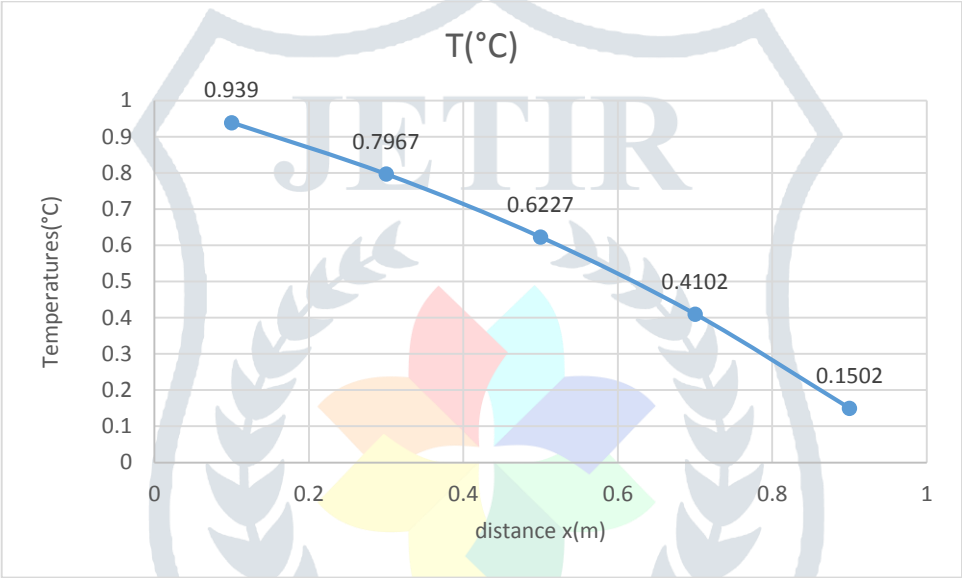
u □□0.1 m/s: F □ρu □□0.1, D □□$\Gamma/\delta x$ □□0.1/0.2 □□0.5

Program for above Numerical problem

```
#include<stdio.h>
#include<conio.h>
#include<math.h>
#define P 1000
int main()
{
float i,n=7,aw,ae,u,rho,gamma,l,T[P],deltax,D,F,tol[P],maxerr,TO[P];
clrscr();
rho=1.0f;
gamma=0.1f;
u=0.1f;
deltax=0.2f;
F=rho*u;
T[1]=1.0f;
T[7]=0.0f;
D=gamma/deltax;
maxerr=1000.0f;
while(maxerr>=1e-06)
{
for(i=2;i<=6;i++)
{
TO[i]=T[i];
}
for(i=2;i<=6;i++)
{
if(i==2)
{
ae=(D-(0.5*F));
aw=(2.0*D)+(0.5*F);
}
else if(i==n-1)
{
ae=(2.0*D)-(0.5*F);
aw=(D+(0.5*F));
}
else
{
aw=(D+(0.5*F));
ae=(D-(0.5*F));
}
T[i]=(aw*T[i-1]+ae*T[i+1])/(aw+ae);
printf("\n%d=%f\t",T[i]);
}
printf("\n the solution is \n");
for(i=2;i<=n-1;i++)
{
printf("\n%f",T[i]);
```

```
}
l=1;
for(i=2;i<=n-1;i++)
{
tol[l]=fabs(T[i]-TO[i]);
l=l+1;
maxerr=tol[2];
if(tol[l]>=maxerr)
{
maxerr=tol[l];
}
printf("\n%f",maxerr);
}
}
getch();
}
```

The graph plotted according to the results availed from the developed C program codes.



Graph 1: this graph plotted between temperatures as (y-coordinate) vs. distance x (m) as (x-coordinate)

V. RESULTS AND DISCUSSION

The output generated from the developed C program codes is implemented for the plot of the graph and these data are then compared with the experimental results to obtain the volumetric heat transfer coefficient. The above developed c program codes is for convection and diffusion problem analysis of heat transfer in porous media.

VI. CONCLUSION

Computational study of convective heat transfer is carried out using c program coding and the output thus generated will be compared with the experimental results. An experimental investigation on the volumetric heat transfer coefficient in porous foams has been carried out using the single blow method. These transient temperature data are compared with the theoretical results to obtain the corresponding volumetric heat transfer coefficients between the foam solid surface and the fluid. The volumetric coefficients thus determined agree well with available sets of experimental data. The method developed in this can be used to explore the interstitial heat transfer within various foams.

VII. ACKNOWLEDGMENT

No great endeavour in any field is possible in solitude. It needs inspiration at every step; I express my deep sense of gratitude to my parents who made this project possible. I would also like to express my gratitude to all who helped me directly or indirectly, I feel a unique pleasure in extending my thanks to various sources for helping me in study in detail. It is quite difficult to mention everyone.

Nomenclature:

c = specific heat (J/kgK)

c_p	=	specific heat at constant pressure (J/kgK)
d_m	=	pore diameter (m)
h_v	=	interfacial heat transfer coefficient (W/m^3K)
k	=	thermal conductivity (W/mK)
T	=	temperature (K)
u_D	=	Darcian velocity (m/s)
ε	=	porosity (-)
ε^*	=	effective porosity (-)
ν	=	kinematic viscosity (m^2/s)
ρ	=	density (kg/m^3)

SPECIAL SYMBOLS

$\langle\phi\rangle^{f,s}$	=	intrinsic average

SUBSCRIPTS AND SUPERSCRIPTS

f	=	fluid
s	=	solid

REFERENCES

[1] X. Fu, R. Viskanta, and J. P. Gore, "Measurement and correlation of volumetric heat transfer coefficients of cellular ceramics", *Exp.Therm. Fluid Sci.*, vol. 17, pp. 285-293, 1998.

[2] K. Kamiuto and S. S. Yee, "Heat transfer correlations for opencellular porous materials", *Int. Commun. Heat Mass Transf.*, vol. 32, pp. 947-953, 2005.

[3] L. B. Younis and R. Viskanta, "Experimental determination of the volumetric heat transfer coefficient between stream of air and ceramic foam", *Int. J. Heat Mass Transf.*, vol. 36, pp. 1425-1434,1993.

[4] K. Ichimiya, "A new method for evaluation of heat transfer between solid material and fluid in a porous medium", *ASME J HeatTransf.*, vol. 121, pp. 978–983, 1999.

[5] Z. Wu, C. Caliot, G. Flamant, and Z. Wang, "Numerical simulation of convective heat transfer between air flow and ceramic foams to optimize volumetric solar receiver performances", *Int. J. HeatMass Transf.*, vol. 54, pp. 1527-1537, 2011.

[6] A. Kopanidis, A. Theodorakakos, E. Gavaises, and D. Bouris, "3D numerical simulation of flow and conjugate heat transfer through a pore scale model of high porosity open cell metal foam", *Int. J.Heat Mass Transf.*, vol. 53, pp. 2539-2550, 2010.

[7] F. Kuwahara, and Y. Fumoto, "The effective pore diameter of a three-dimensional numerical model for estimating heat and fluid flow characteristics in metal foams", *Open Transport PhenomenaJ.*, In Press.

[8] C. Y. Liang, and W. J. Yang, "Modified single blow technique for performance evaluation on heat transfer surfaces", *Trans. ASME J.Heat Transf.*, vol. 97, pp. 16-21, 1975.

[9]N. Wakao and S. Kaguei, *Heat and mass Transfer in packed beds*,Gordon and Breach Science Pub: NY, 1982.

[10]H K Versteeg and W Malalasekera,An Introduction to Computational Fluid Dynamics, Second Edition, Pearson Publisher ,2007, page no.115-176 and 243-266.