International Journal of Applied Engineering Research ISSN 0973-4562 Volume 10, Number 21 (2015) pp 42428-42433
© Research India Publications. http://www.ripublication.com

Thermal And Fluid Flow Analysis Of Washing Machine Using CFD

Saswati Nayak[1]

PG Student, Department of Mechanical Engineering, AITAM, Tekkali, Email:saswatinayak2010@gmail. com

K. Mohan Laxmi[2]

Assistant professor, Department of Mechanical Engineering, AITAM, Tekkali. Email:lakshmi100988@gmail. com.

M. V. Ramana[3]

Associate professor, Department of Mechanical Engineering, AITAM, Tekkali. Email:mvramana6028@gmail. com.

Abstract
A computational model to predict the unsteady, multiphase free-surface flows inside a washing machine has been developed in this work. The prevailing multi-phase flow computations are performed using the Volume of Fluid (VOF) method. Two phase (air-water) and three phase (air-water-detergent) simulations are conducted. Wash tub is held stationary and the agitator is given unidirectional rotary motion. Governing equations for continuity, momentum, swirl velocity, and volume fraction are solved. These equations are first integrated about a control volume and the resultant algebraic equations are solved numerically. Unsteady computations are performed using implicit scheme. Results obtained using two-phase simulations show that the velocity inside the washing machine fluctuates heavily sloshing up and down. The location of the free-surface was captured as a function of time. The free-surface was found to be going up and down as the agitator is rotating. Three-phase simulations indicate that the detergent mixed well with water in about 0. 2 s. and also observe the temperature variations and heat flow. The proposed computational model could be used as a tool to design better and efficient washing machines.
Keyword:-The unsteady and free surface flow was taken inside the washing machine in volume of flow method. Both two phage and three phase simulations are conducted. Results are obtained inside the washing machine when velocity fluctuates heavily. The location of free surface is captured as function of time.

1. Introduction:

Free-surface flows are encountered in several applications in our life ranging from simple flow inside a rotating bowl to as large as the flow of a river. Free-surface flows occur whenever a liquid comes in contact with a gas and there is a flow of any one or both the fluids. The liquid-gas interface exhibits a severe surface force (due to surface tension) and causes complexity in the flow structure. Several studies have been performed in the past to understand the physics of such flows. Recent developments in computational fluid dynamics (CFD) enabled the researchers to simulate almost all kinds of problems that can occur in real situations. Particularly, free-surface flows occurring in internal flows are of great importance. Flow inside a washing machine (WM) is an example of internal free surface flows. Understanding the flow structure and mixing pattern allows us to design efficient and economical washing machines.
The objective of the present study is to develop a CFD model that can predict the unsteady free-surface flows inside a washing machine. The development of the theoretical model is described in the next section. Solution procedure, grid generation and numerical techniques are given in the following section. Results obtained using the proposed model is described next followed by the conclusions.

1. 1 Continuity equation

When a fluid is in motion, it must move in such a way that mass is conserved. To see how mass conservation places restrictions on the velocity field, consider the steady flow of fluid through a duct (that is, the inlet and outlet flows do not vary with time). The inflow and outflow are one-dimensional, so that the velocity V and density are constant over the area A.
Flow rate=area (A) x velocity (V)

1. 2 Momentum equation

A representative velocity for all the phases is obtained by solving a single momentum equation throughout the domain. The resultant velocity is then shared among the phases. This is linked with volume fraction through density and viscosity.

1. 3 Swirl velocity

Swirlvelocity Is the tangential velocity flow of fluid i. e. the flow happens in the angular direction.

1. 4 Volume fraction

The volume fraction Ψ is defined as the volume of a constant v1 divided by the volume of all the constraints of the mixture v prior to mixing.

2. Review of literature:-

Ira H. Abbott& Albert E. Von Doenhoff [1] published a book in order to meet the continuing demand for a concise compilation of the subsonic aerodynamic characteristics of modern NACA wing sections together with a description of their geometry and associated theory. these wing sections, or their derivatives, continue to be the most commonly used ones for airplanes designed for both subsonic and supersonic

speeds, and for applications to helicopter rotor blades, propeller blades, and high performance fans.

Schilting [2] used artificial techniques to control the boundary layer and was successful in controlling the boundary layer over the hull of the vehicle, by various techniques which has been developed and brought by him. An appropriate shape is a known means for extending the laminar boundary layer flow to greater length. The approach to laminarisation is called boundary layer stabilization by pressure gradient or briefly the laminar aerofoil effect. Thus the stabilization techniques are used to control boundary layer formation. Transition from laminar to turbulent flow can also be delayed by the use of suitable shaped bodies, Schilting. Boundary layer theory gives an answer to the very important question of what shape a body must be given in order to avoid this detrimental separation.

Sighard f. hoerner [3] briefly described the theoretical and information of skin friction drag and pressure drag over various axisymmetric bodies. He also explained the importance of boundary layer flow, flow separation effecting the drag and lift of the body. He also gave brief description related to variation of drag due to change in nose profiles and also their effect on air ships at various speeds.

Md. Akhtaruzzaman Sarkar and David G. Rhodes [4] made measurements of the free surface profile in a laboratory channel housing a broad crested weir at the maximum flow rate of the facility. The experimental arrangement was then numerically modelled by CFD commercial software (FLUENT), using the standard K-E turbulence model and wall function, in conjunction with the volume of fluid free surface model. Predicted results agreed closely with experiment near the critical section on the weir and in the super-critical flow downstream of the weir. However, the CFD under-predicted the upstream water level.

MajumdarandB. N. Rajani [5]have evaluated grid generation for arbitrary 3 –D configuration using a differential algebraic hybrid method. Computational Fluid Dynamics (CFD) is relatively new field; it refers to the computational solution of fluid dynamics problems. CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. Major steps involved in grid generation, flowsolver, output.

H. yasukawa [6] applied a rankine source method to the shallow water and channel problems, and calculations were made of free –surface flow around a ship moving in calm water and her wave making resistance. The calculated results were compared with experiments and calculations by conventional linerized analytical method. It was shown that the Rankine source method promises an improvement in predicting the free surface flow around a ship in shallow water than the conventional linerized analytical method.

Hirt, c. w and Nichols, b. d [7], used a simple method to approximate free boundaries in finite difference numerical simulations, based on the concept of a fractional volume of fluid. The flexibility and efficiency of the method is demonstrated by means of an incompressible hydrodynamics code that uses the volume of fluid technique to track free fluid surfaces. Sample calculations with the nw\ew code show that it works well for wide range of complex problems.

Launder, B. E., and Spalding, D. B. [8] The earlier turbulence models, k-ε model focuses on the mechanisms that affect the turbulent kinetic energy. The mixing length model lacks this kind of generality. The underlying assumption of this model is that the turbulent viscosity is isotropic, in other words, the ratio between Reynolds stress and mean rate of deformation is the same in all directions.

S. V Patankar [9] His most important contribution to the field of CFD is the simple algorithm. Patankar is the author of a book in computational fluid dynamics titled Numerical Heat Transfer and Fluid Flow which was first published in 1980. This book has since been considered one of the groundbreaking contributions to computational fluid dynamics due to its emphasis on physical understanding and insight into the fluid flow and heat transfer phenomena. He is also one of the most cited authors in science and engineering.

3. Methodology:-

3. 1Problem definition and solution

3. 1. 1 Grid generation:

A 2D axisymmetric geometry of size 0. 299 X 0. 842 m is considered in the present work. The external dimensions correspond to a commercial washing machine P1405J. A grid of triangular elements is generated using a commercial software GAMBIT is the computational grid employed in this study.

3. 1. 2 Solution procedure

Governing differential equations are integrated about each control volume. This process yields a set of algebraic equations that conserve a quantity on a control-volume basis. These algebraic equations are then solved numerically. Pressure and velocity coupling is achieved by using SIMPLE algorithm. Momentum, swirl, and turbulence quantities are discretized using the Power law scheme.

Unsteady calculations are performed using Explicit-time marching scheme with an automatic step refinement option. The tabular form is shown in table 1.

The under-relaxation factors and the initial guess values used in this study are summarized in Table 2 phase (air and water) and in table 3 for three phases (air, water and detergent).

The various physical boundaries are mapped into relevant computational boundaries in table 4.

3. 1. 3 Modelling outline in FLUENT

The mathematical model described in the previous sections can be solved using FLUENT. The steps involved in the modelling are outlined below:

3. 1. 3. 1Steps in GAMBIT:

1. Create the washing machine geometry
2. Generate mesh
3. Define boundary and continuum types
4. Export mesh

3. 1. 3. 2 Steps in FLUENT:

1. Read the mesh files into FLUENT
2. Setup Multiphase model
3. Define boundary conditions

4. Specify wall rotary motion
5. Patch initial water and detergent levels
6. Solve
7. Post processing

4. Concepts of computational fluid dynamics and fluent:-

Fluid dynamics is a science, which deals with the motion of the fluids, forces caused for the fluid motion and dynamics behaviour of the fluid under these forces. There are two approaches in the fluid dynamics; experimental fluid dynamics and theoretical fluid dynamics. Computational fluid dynamics constitutes new "third approach" In the philosophical study and development of the whole discipline of fluid dynamics. The advent of the high-speed digital computer combined with the development of accurate numerical algorithms for solving physical problems on these computers has revolutionized the way we study and practice fluid dynamics today. It has introduced a fundamentally new third approach in fluid dynamics –the approach of computational fluid dynamics.

Finally, we note that computational fluid dynamics is a common place enough today that the acronym CFD is universally accepted for phrase "computational fluid dynamics". This acronym is used throughout the project.

4. 1 computational fluid dynamics

Computation fluid dynamics mainly depends upon three fundamental governing equations of fluid dynamics

- Mass conservation (Continuity equation)
- Momentum Conservation (Navier-Stokes Equation)
- Energy Conservation (Energy equation)

The various complex and interesting flows result from the solution of these equations subjected to different physical boundary conditions. The numerical solution of these equations subjected to the boundary conditions can be obtained by the use of the following methods:

1. Finite Difference Method (FDM)
2. Finite Element Method (FEM)
3. Finite Volume Method (FVM)

There are a variety of other methods but the three above are mostly used in the fields of CFD research. In this work, we have used the commercial software, "FLUENT" which uses the FVM. In the following paragraphs, the FVM is very briefly described.

Finite Volume method often called the Control Volume Method is formulated from the inner product of the governing differential equations with a unit function. This process results in the spatial integration of the governing equations. The integrated terms are approximated by either finite differences or finite elements discretely summed over the entire domain.

4. 2. Discretization

The discretization procedure approximate the space and time derivatives of the flow variable at each node to algebraic functions of the values of the variables at the given node and a selection of its neighbours is done through the four principal strategies that can be adopted to do this.

These are:

1. The finite difference method
2. The finite volume method
3. The fin ite element method

4. 2. 1 Finite difference method:

This is the most direct and the most obvious of discrediting any differential equation governing a variable ϕ. The variation of ϕ is approximated by some function of distance in the coordinate directions and is fitted to the selected nodes. Here a forward differencing approximation has represented the time difference and the spatial derivatives by cantered differencing schemes are possible such as upwind differencing approximation. This can be divided into two types.

4. 2. 1. 1 Explicit Method:

By definition in an explicit approach each differential equation contains only one unknown and therefore can be solved explicitly for this unknown in straightforward method.

4. 2. 1. 2 Implicit Method:

An implicit approach is one where the unknowns must be obtained by means of a simultaneous solution of difference equations applied at all grid points arrived at a given time level.

4. 2. 2 Finite volume method:

Finite volume methods are essentially a generalization of the finite difference method but use the integral form of the governing equations of flow, rather than the differential form. This gives greater flexibility in handling complex domains, as the finite volumes need not to be regular. A finite volume method on a non-staggered non-orthogonal grid, applicable to complex geometries. This approach has some problems, the main one being that the staggered grid with non-coinciding pressure and velocity grid points traditionally used by finite difference solver to avoid pressure velocity de-coupling is difficult and computationally expensive to use in complex geometries.

4. 2. 3 Finite element method:

The finite element method achieves discretization of the governing equation in two approximation stages. First an assumption is made about the spatial variation of ϕ within each element by means of a 'trial function'. This is essentially a local interpolation applied over an element, linking the solution at apoint within the element to the nodal values. The second stage uses a weighted residual method

4. 3 CFD flow:

The fundamental method for creating a CFD model can be represented as a "pipeline" of six steps as illustrated in Figure below. The process begins with the creation of an assembly of one or more parts (geometric shapes) that reflect the design under consideration. Because CFD deals with the flow through or around solid objects, the CFD model is built upon the inverse of the assembly and is referred to as the "Flow Volume Extract". This Flow Volume Extract is meshed tospecification, a simulation is performed, and finally, the results of the simulation are analyzed.

4. 4 Introductions to fluent:

FLUENT is a state-of-the-art computer program for modelling the fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility solving your flow problem with un-structured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2-D triangular/ quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge, and mixed (hybrid) meshes. FLUENT also allows you to refine your grid based on the flow solution.

4. 5 Overview of using fluent:

FLUENT uses unstructured meshes in order to reduce the amount of time you spend generating meshes, simplify the geometry modelling and mesh generation process, model more-complex geometries than you can handle with conventional, multi-block structured meshes, and let you adapt the mesh to resolve the flow-field features. FLUENT can also use body-fitted, block-structured meshes (e. g., those used by FLUENT 4 and many other CFD solvers). FLUENT is capable of handling triangular and quadrilateral elements (or a combination of the two) in 2D, and tetrahedral, hexahedral, pyramid, and wedge elements (or a combination of these) in 3D. This edibility allows you to pick mesh topologies that are best suited for your particular application. You can adapt all types of meshes in FLUENT in order to resolve large gradients in the flow field, but you must always generate the initial mesh (whatever the element types used) outside of the solver, using GAMBIT, T Grid, or one of the CAD systems for which mesh import filters exist.

4. 6 Problem solving steps:

Once you have determined the important features of the problem you want to solve, you will follow the basic procedural steps shown below.

1. Create the model geometry and grid.
2. Start the appropriate solver for 2D or 3D modelling.
3. Import the grid.
4. Check the grid.
5. Select the solver formulation.
6. Choose the basic equations to be solved: laminar or turbulent (or in viscid), chemical species or reaction, heat transfer models, etc. Identify additional models needed: fans, heat exchangers, porous media, etc.
7. Specify material properties.
8. Specify the boundary conditions.
9. Adjust the solution control parameters.
10. Initialize the flow field.
11. Calculate a solution.
12. Examine the results.
13. Save the results.
14. If necessary, refine the grid or consider revisions to the numerical or physical model.

5. Discussion:-

5. 1 Grid sensitivity analysis:-

In order to demonstrate the gird independent nature of the results, three girds of different sizes have been generated. The following grid sizes are considered: (a) 3032 nodes, (b) 11764 nodes, and (c) 46328 nodes. The axial velocity variation at the bottom of the washing machine with respect to flow time was captured and the results are plotted in Fig 1.

5. 2 Validation of the model:

The proposed numerical model has been qualitatively validated with those results obtained in a FLUENT and the models the free-surface flow of water inside a rotating bowl and with no central disk or agitator. In the present study, an agitator was defined to model the flow inside a washing machine and a unidirectional motion was given and the tub (outer region) was kept stationary. However, it is appropriate to compare the results qualitatively. The parameters considered for comparison is given in Table 5.
For the present model of the flow is shown in fig 2.

5. 3 Results of two phase simulations:

In this case, bottom 1/3 of the bowl was filled with water. A rotational speed of 3 rad/s was given to the agitator. The free-surface boundary was captured for the first 0. 2sec and 2sec. The water volume fraction contours have been obtained and presented in Figs 3 (0. 2 s) and fig 4 (2s). Results clearly show that the free-surface boundary is affected by the rotary motion of the agitator. According to Figs 3 and 4, the free-surface boundary inside the washing machine goes down until 0. 3 s and then starts to go up and returns to its original boundary. This process then repeats showing that there are severe oscillations present inside the washing machine.

5. 4 Results of three phase simulations:

The flow and mixing pattern inside the washing machine when filled with water and detergent are obtained. This would allow us to understand the free-surface boundaries as well as the effectiveness of the mixing. Liquid ammonia has been chosen to represent the detergent.
In this case also, bottom 1/3 of the bowl was filled with water. An initial volume fraction of 0. 05 is given for detergent. A rotational speed of 3 rad/s was given to the agitator. The free-surface boundary was captured for the first 2 s. The detergent volume fraction contours have been obtained and presented in Figs 5(0. 2 s) and fig 6 (2s).
The free-surface between water-detergent mixture and air is affected by the rotary motion of the agitator. According to Figs. 5 and 6 the free-surface boundary inside the washing machine first goes down until 0. 3 s and then starts to go up and returns to its original boundary. This process then repeats showing that there are severe oscillations present inside the washing machine. It is also evident that the detergent is able to mix well with the water and air produce a uniform environment for washing.
.

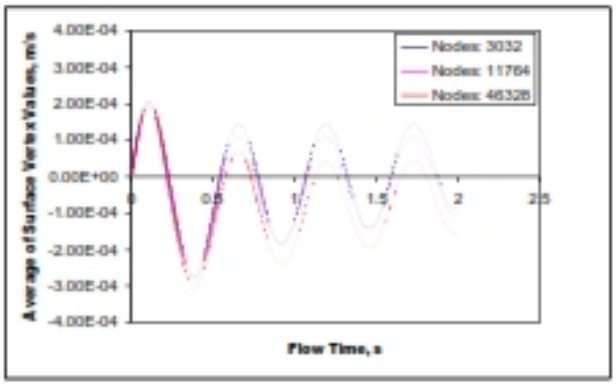
Sl. No.	Parameter	Method
1.	Pressure	Body-force weighted
2.	Momentum	Power law
3.	Swirl velocity	Power law
4.	Turbulent kinetic energy	Power law
5.	Turbulence dissipation rate	Power law
6.	Pressure-velocity coupling	SIMPLE
7.	Unsteady formulation	Explicit scheme

Sl. No.	Parameter	Value
1.	Pressure	0.3
2.	Density	1.0
3.	Body forces	1.0
4.	Momentum	0.7
5.	Swirl velocity	0.9
6.	Volume fraction	0.2
7.	Turbulent kinetic energy	0.8
8.	Turbulence dissipation rate	0.8
9.	Turbulent viscosity	1.0

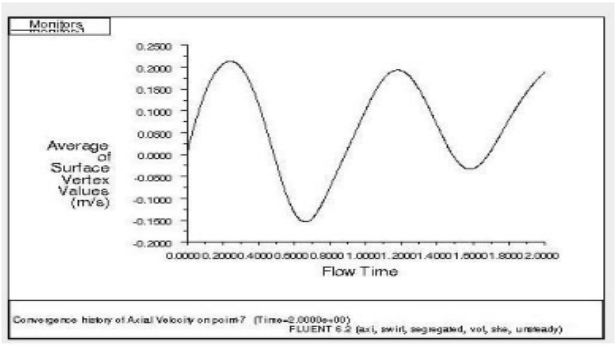
Sl. No.	Parameter	Value
	Initial Guesses	
1.	Gauge pressure, Pa	0.0
2.	Axial velocity, m/s	0.0
3.	Radial velocity, m/s	0.0
4.	Swirl velocity, m/s	0.0
5.	Turbulent kinetic energy, m^2/s^2	0.0
6.	Turbulence dissipation rate, m^2/s^3	0.0
	Patch Values	
1.	Water volume fraction	0.95
2.	Detergent volume fraction (for 3-phase)	0.05

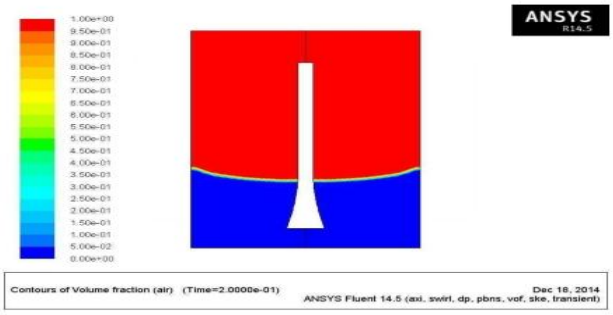
Boundary conditions

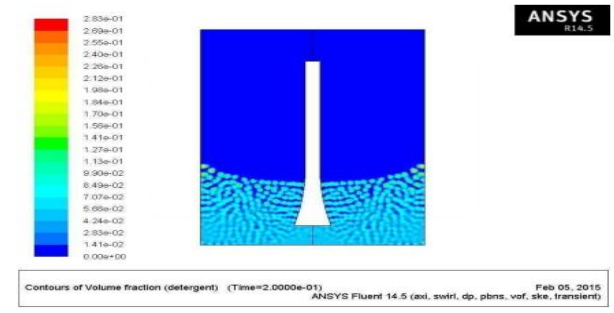
Sl. No.	Physical Boundary	Computational Mapping
1.	WM lid	Pressure inlet
2.	WM bottom	Wall
3.	WM tub	Wall
4.	Agitator	Rotating wall

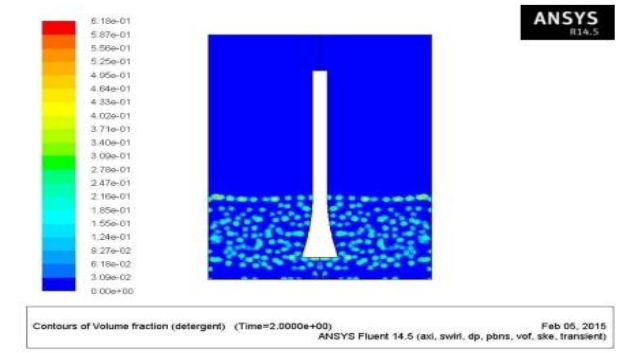


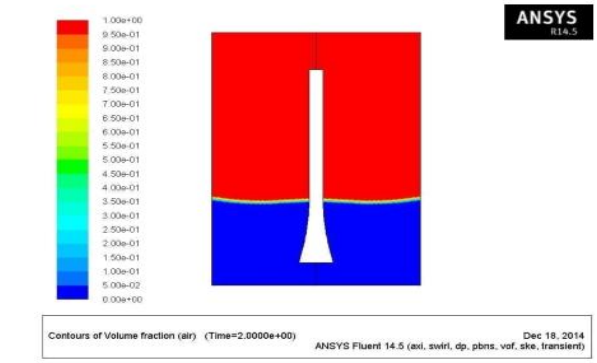
Sl. No.	Parameter	Present Model
1.	Geometry	2-D Washing Machine
2.	Size	0.299 x 0.842 m
3.	Rotation type	Angular, 3 rad/s
4.	Rotating part	Agitator
5.	Agitator Availability	Yes
6.	Direction of Gravity	Negative X-Direction











6. Conclusions:-

The following conclusions could be drawn from the present work:

(1) A computational model to predict the flow and mixing pattern inside washing machine has been developed. The proposed model is capable of capturing the unsteady free-surface boundaries.

(2) Using this model, free-surface boundaries in water-air and water-air-detergent interfaces in a washing machine have been predicted.

(3) Results show that the water in the washing machine sloshes up and down and would reach a steady state eventually.

(4) This computational model could be used as a tool in designing better and efficient washing machines.

7. References:-

1. Ira H. Abbott & Albert E. Von Doenhoff:-Theory of Wing Sections, including a summary of airfoil data. Dover Publications. ISBN 978-0-486-60586-9.
2. H. Schlichting (Author), K. Gersten (Author) :- Boundary-Layer Theory Hardcover – March 22, 2000
3. Sighard f. Hoerner, Dr. Sighard F. and Borst, Henry V. :-Fluid-Dynamic Lift, Hoerner Fluid Dynamics, Bricktown New Jersey, 1975.
4. Md. Akhtaruzzaman Sarkar and David G. Rhodes:- Open Journal of Soil Science, 2014, 4, 26-35 Published Online January 2014.
5. Majumdar and B. N. Rajani Department of Mathematical and Computational Sciences, National Institute of Technology, Karnataka, Surathkal 575 025, India
6. H. yasukawaInternational Conference on Numerical Ship Hydrodynamics**.**
7. Hirt, C. W, and Nichols, B. D., (1981) "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, " Vol. 39 pp. 201-225. Fluent 6. 0 Documentation, FLUENT Inc.
8. Launder, B. E., and Spalding, D. B., (1974), "The Numerical Computation of Turbulent Flows, " Computer Methods in Applied Mechanics and Engineering, Vol. 3, pp. 269-289.
9. Patankar, S. V. (1980), Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Co., NewYork.