International Journal of Application or Innovation in Engineering & Management (IJAIEM)
Web Site: www.ijaiem.org Email: editor@ijaiem.org, editorijaiem@gmail.com
Volume 2, Issue 5, May 2013 ISSN 2319 - 4847

COMPUTATIONAL INVESTIGATION OF FREE SURFACE FLOW AROUND A SHIP HULL

Katuri Samarpana, Ajay konapala, Duvvada Ramesh

M.V.G.R.College of Engineering / Mechanical, Vizianagaram, India

ABSTRACT

Ship hydrodynamics present many challenges due to complex geometry, environment, and operating conditions, which results in many complex physics and modeling issues. This is commonly studied through experiments using LAHPMM in a towing tank and experiments in a sea keeping and maneuvering basin. Recently hydrodynamicists have begun to venture into computational prediction of hydrodynamic behavior of surface ships. Free surface phenomenon around a ship hull plays an important role in its resistance. Wave making resistance comes from the very presence of free surface. Therefore its accurate prediction is very essential for ship design. Experimental methods have been traditionally used to predict the ship resistance. Though expensive, experimental methods provide a good solution to this problem. CFD methods, which are getting matured fast, desired for routine computation along with continuous reduction in the cost computation, provides another way to study such phenomenon. The flow problem to be simulated is rich in complexity and poses many modeling challenges because of the existence of breaking waves around the ship hull involving two-phase flow, and because of the resolution of thin turbulent boundary layer. Research is go in still globally to establish better results in this area. flow through the ship hull is computed using a finite volume commercial code, FLUENT 6.2

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. Simulating fluid flow around ship hull is useful for a no of reasons, including exploring the effect of hull shape on drag and understanding the dynamics in sea. Keeping exercise for computational fluid dynamics to see fully accepted in the marine industry for these applications. CFD fluent need to deliver accurate, efficient simulations in the steady state and transient situational

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 are described next followed by the conclusions.

2. GOVERNING EQUATIONS

Flow simulations in ship involve unsteady free-surface flows with multi-phase calculations. The Volume of Fluid (VOF) approach [1] has been adopted and a single continuity equation for volume fraction for (n-1) phases is solved.

Continuity equation

Continuity equation for volume fraction for all the secondary phases is given below:

$$\frac{\partial \alpha_q}{\partial t} + \vec{v} \cdot \nabla \alpha_q = 0 \qquad (1)$$

and the primary phase is calculated using:

$$\sum_{q=1}^{n} \alpha_q = 1 \qquad (2)$$

The transport coefficients are evaluated as follows:

$$\Gamma = \sum \alpha_q \Gamma_q \qquad (3)$$

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 [2].

$$\frac{\partial}{\partial}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v}\vec{v}) =$$
$$-\nabla p + \nabla \cdot [\mu(\nabla \vec{v} + \nabla \vec{v}^{T})] + \rho \vec{g} + \vec{F} \qquad (4)$$

Turbulence
The rotary motion of agitator induces turbulence inside the washing machine. The turbulence is modeled using the standard k-ε turbulence model [3]. The governing equations used are similar to those found in Ref. 3.

3. SOLUTION METHODOLOGY

Grid generation
ANSYS ICEM is used for the high quality hexa mesh generation. To generate the structured grid with hexahedral cells commercially available grid generation software ANSYS ICEM CFD V 12.1 is used. Free surface computations require fine mesh in the region around the free surface so as to capture the free surface flow, which can be observed in figures. Mesh on the ship hull is shown in fig 1.Total number of cells in the fluid region: 1427564. Minimum mesh quality – 0.27

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 [4]. 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 Courant number is taken as 0.25. Various discretization schemes used in the study are summarized in Table 1.

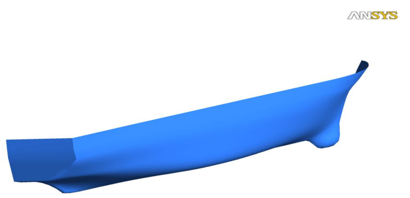
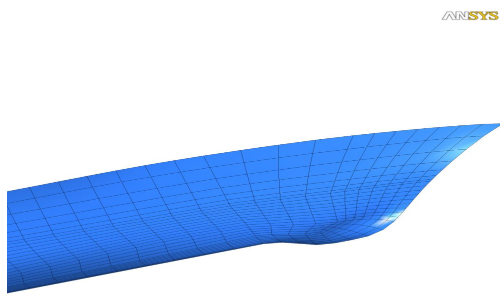


Fig. 1 Geometry of free surface body after healing.



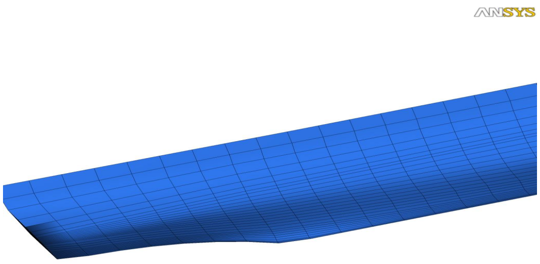


Fig. 2 Mesh at the free surface location

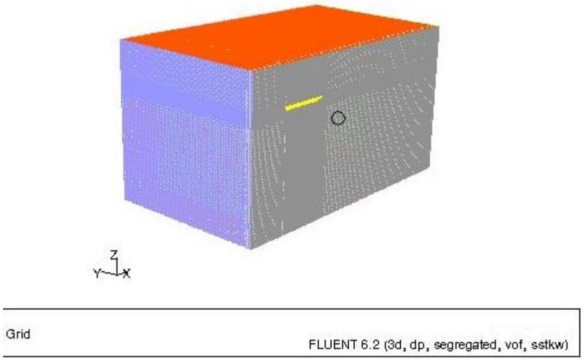
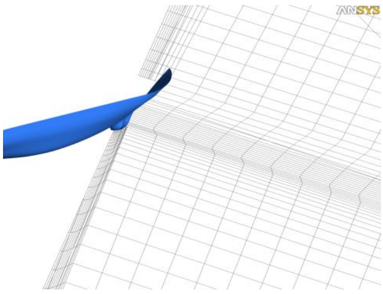
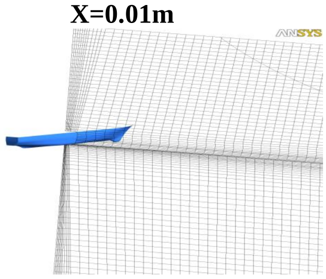


Fig 3 Complete meshing of ship hull with domain.

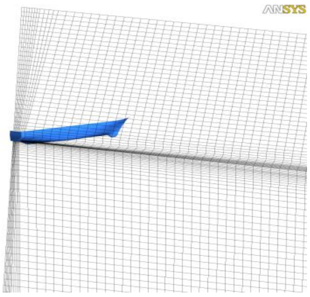
MESH AT DIFFERENT VERTICAL CUT PLANE



X=0.01m



X=0.07m



X=0.135m

Fig 4 Mesh at different cut plane

Table 1 Solver settings:

Solver Settings	
Slover	Implicit, Segregated 3D
Pressure Link	PISO
Pressure	Body Force Weighted
Convective Fluxes	QUICK
Diffusive Fluxes	Central Difference
Turbulent KE	Second Order Upwind
Turbulent Dissipation rate	First Order Upwind
Multiphase model	Volume of Fluid
Turbulent model	Reynolds Stree model
Near Wall Treatment	Standard Wall Functions

Table 2 Fluid properties:

For Air :	
Fluid Density	1.225 kg/m^3
Viscisity	1.7894e-05 kg/m-s
For Water	
Fluid Density	998.2 kg/m^3
Viscisity	0.001003 kg/m-s

Table 3 Phases:

Primary Phase	Air-Phase
Secondary Phase	Water-Phase

Boundary conditions

- Inlet boundary condition is specified at the front of domain
- Top and side of the domain is refined as free slip walls
- Bottom of the domain defined as no slip wall
- A pressure outlet condition is defined at the rear of the domain.
- Hull is defined as no slip wall condition

The FLUENT V6.3 Code is used to solve the 3D incompressible flow. The parallel version of FLUENT simultaneously computes the flow equations using multiple processors. Here, the problem is analyzed using a single processor on a 8 processor based Sgi-Altix machine as there is no restriction on the computational speed.

Gas-liquid interface interpolation scheme
To obtain the properties at the interface, the Geometric Reconstruction procedure is adopted [2]. In this approach, when a control volume is completely filled with one phase or the other, the standard interpolation scheme is used. If an interface control volume is partially filled, the properties are obtained by assuming a piece-wise linear relationship.

APPROACH:

- Half symmetry model is being used.
- An external domain is defined around the hull with the following dimensions:
- 1.5 ship length at the front of the ship
- 3 ship lengths at the rear of the ship
- 2 ship length at the side of the ship
- 3 ship length at the bottom of the ship

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

Steps in CATIA and ANSYS ICEMCFD:

1. Create the ship geometry and farfields
2. Generate mesh

3. Define boundary and continuum types
4. Export mesh

Steps in FLUENT:

1. Read the mesh file 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. Postprocessing

4. RESULTS AND DISCUSSION

Results for Flow around the Ship:

Analysis is carried out for flow around the ship hull, when submerged to the design draught level. The computations are performed for a scaled down model. Model scale factor is 1000. Flow around the Ship is solved for unsteady and turbulent condition at a model speed of 0.569 m/s. This speed corresponds to the Froud number of the actual ship moving at design speed. Wave making resistance around the hull is calculated and compared with the experimental values.

The ship hull (scaled model) is moving at a speed of 0.569m/s. Whereas unscaled modal, the speed is 18m/s.

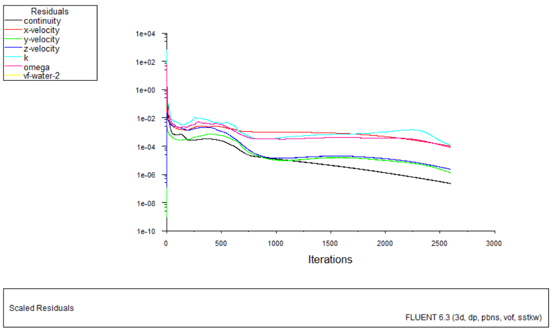


Fig 5 Monitor for momentum, pressure and turbulence equations

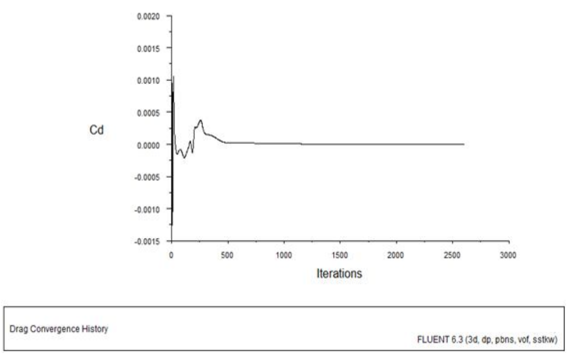


Fig 6 Monitor for drag coefficient

Final Drag Coefficient Value: 0.0001896

DRAG (Total Resistence) = coff.drag * ½ * density * wetted Area * sq of velocity

Rt = 0.0001896 * 0.5 * 1000 * 2042 * (0.569 * 0.569)

= 62.67N (Scaled)

= 62.76*1000 = 627600

= 672.6KN

ISO Surface of water volume fraction 0.5 colored with the height

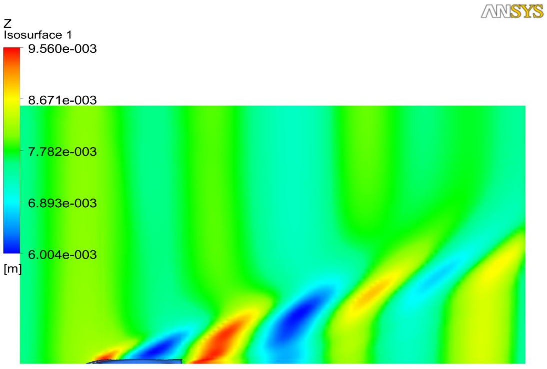
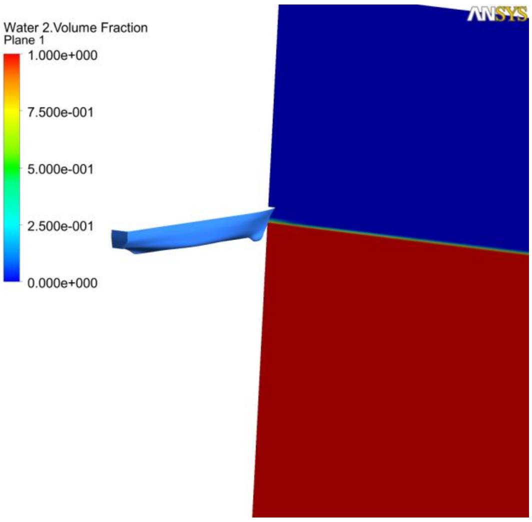
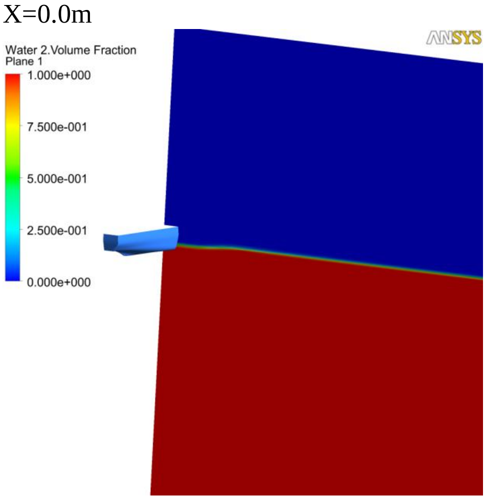


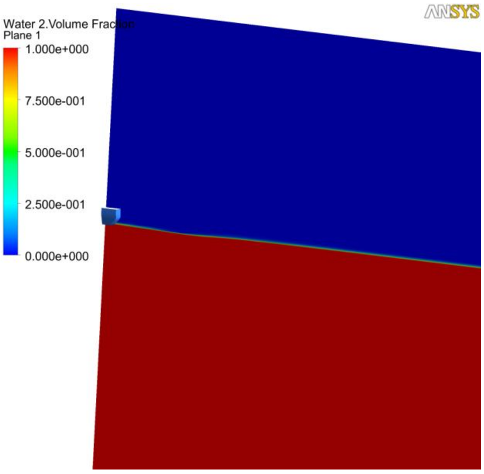
Fig 7 ISO Surface of water volume fraction 0.5 colored with the height



X=0.0m



X =0.08m



X=0.135m

Fig 8 Location of free surface at different cut plane.

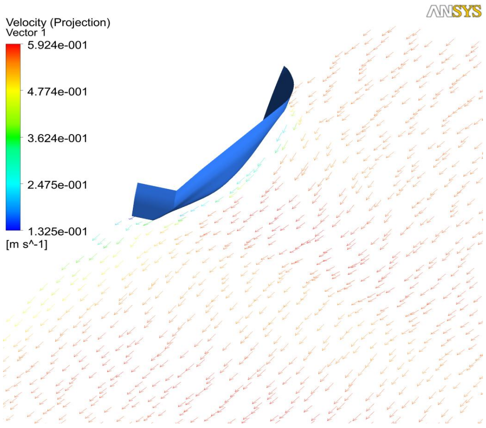


Fig 9 Tangential projection of the water velocity at free surface

Theoretical Estimation of resistance

To estimate the total resistance of the given model.
The problem at hand is preliminarily solved using empirical relations based on Holtrop-Mennen method
The total resistance of a ship has been sub divided into

$R_{Total} = R_f (1+K_1) + R_{APP} + R_W + R_B + R_{TR} + R_A$

Where

R_F= Frictional Resistance according to ITTC 1957 formulae
1+k1= Form Factor
R_{APP}= Resistance of appendages
R_W= Wave making resistance
R_B= Additional pressure resistance of the bulbous bow near the water surface
R_{TR}= Additional pressure resistance of the immersed transform stress
R_A= Mode-ship correlation resistance
In the present case frictional and wave making resistance are estimated, the other components require additional data.

Main particulars
LOA-147.34 M
Depth= 14.85M
Breadth=17.3 M
Draft=4.5 M
LWL @ 4.5M=135 M
Speed v- 18 m/s
Volume of ship @4.5 m draft is
∇ = 5087.712 m^3.

Calculation of RF:

$\mathbf{R_F = C_F \ X \ ½ \ Pv^2.s}$

Where cf= coefficient obtained from ITTC live

P= Density of the sea water= 1021 KG/M^3 @ 30^0 c

S= wetted surface= 2041.607m^2 (obtained from the model)

V= Speed of the ship (M/s) = 18M/s

$\mathbf{C_F = 0.075/ (Log_{10}R_n - 2)^2}$

R_n= Reynold's number = VL/ν

ν=Kinematic viscosity=0.84931 x 10^{-6} m^2/s.

V=18 m/s

L=135m

ν=0.84931 x 10^{-6} m^2/s.

R_n=18x135/0.84931 x 10^{-6} = 2.861x10^9.

C_F = 0.001349.

Therefore frictional resistance

R_F= $\mathbf{C_T x1/2xPxSxV^2}$.

=0.001349x0.5x1021x2041.607x18^2

=**455.826 KN.**

Ware making resistance

$\mathbf{R_w = c_1c_2c_5 \nabla \ pg \ exp \ (m_1f_n^d + m_2cos(\lambda fn^{-2})}$.

Block coefficient C_B = ∇ /LBT.

= 5087.712/147.34x17.3x4.5 =0.4435

CM=A_M/BxT= 59.49/17.3x4.5 = 0.764

Long prismatic coefficient C PL = CB/CM= 0.4435/0.764 = 0.5804.

C1= 2223105 c7 $^{3.78613}$ x exp{-(L/B)$^{0.80856}$ x(1-CwP)$^{0.30484}$x(1-C_p- 0.0225Wb)$^{0.6367}$ x (LR/B)$^{0.34574}$ x (100∇ /L^3) $^{0.16302}$ =13 ^{0}.

C_1= 2223105x (0.117416)$^{3.78613}$x (4.5/17.3)$^{1.07961}$ x(90-13)$^{-1.37565}$

= 2223105x3.005x10^{-4}x0.23367x2.5623x10^{-3}

C_1=0.399

C_2= exp(-1.89√c3)

C_3=0.56.$A_{BT}^{1.5}$/{BT(0.31(0.31√A_{BT}+ T_F-G_B)}

Hb=2.2

TF=4.5

ABT=59.49

B=17.3

T=4.5

C_3= 0.56(59.49)$^{1.5}$/17.3x4.5x(0.31x√59.49+ 4.5-2.2)

= 256.95/365.196

=0.7036

Therefore C2= exp(-1.89x√59.49)

C2=0.2049

C_5= 1-0.8A_T/(BTC_M)

A_T Transfer area

A_T = 4.106

C_5 = 1- ((0.8x4.106)/(17.3x4.5x0.764))

=1-(3.2848/59.4774)

$C_{5=}$ 0.9448

M_1= 0.0140407x (L/T)-1.75254x∇ $^{1/3}$xL-4.79323xB/L-C_{16})

Where C_{16}= 8.07981C_P- 13.8673C_p^2+6.984388C_p^3 =8.07981x0.5804-13.8673x $(0.5804)^2$+6.984388x $(0.5804)^3$
= 4.689521724-4.671396+1.36555
=1.3868

M_1= 0.0140407x (135/4.5)-1.75254x$(5087.712)^{1/3}$x135-4.79323x(17.3/135)-C16

= 0.421221-0.22328-0.61424-1.38368
=m_1= -1.7999

λ= 1.4$C_1$6C_P-0.03x (L/B)
=1.446x0.5804-0.03(135/17.3)
=0.83925-0.234104
=0.60515

F_n=V/√(gl)=18/(√(9.8x135)=0.49487

M_2= c_{15} cp^2exp $(-0.1xF_n^{-2})$
=-1.69385x0.5804^2xexp (-0.1x $(0.49487)^{-2}$)

M_2=-0.3793

R_W= c1xc2xc5x∇ pg exp {$m_1F_n^d$+$m_2$cos (λF_n^{-2})}

=0.399x0.2049x0.9448x5087.712x10210.7x9.8xexp [-1.7999x $(0.49487)^{-0.9}$+ (-0.3793).cos0.60515x $(0.4948)^{-2}$]

=90796.94N

R_w= 90.796KN

Total resistance [frictional and wave making]

R_T=455.826+90.796=546.622KN

The above calculations are made for prototype and the same can be scaled down to model using linear scale ratio i.e. L_s/L_m= λ

$R_m/R_S=1/\lambda^3$ since $R_m=Rs/\lambda^3$

Here λ=1000

R_s=546.622KN

$R_m=R_s/\lambda^3$ = 564.622/10^9 = 0.546x10^{-3}N.

Note: if the other components are to be estimated the area of appendages is required. However the total resistance shall go up by another 150KN if the appendage and model correlation is also added.

Results when the model speed is at 0.569m/s corresponding to Ship speed of 18 m/s.	**Exp Results**	**CFD Results**	**Analytical result** Hiltop-Menon
Total Resistance (KN)	**636.32**	**672.7**	**546.62**
Friction Resistance (KN)	**524.12**	---------	**455.82**
Wave Making Resistance (KN)	**112.20**	---------	**90.79**

Table 4 Comparison between experimental, Analytical and computational values.

The above table shows experimental, Analytical and computational values of total resistance are closely matching. The total resistance on the hull is dependent on the wave pattern immediately around the hull. The wave pattern on the hull is captured computationally.

5. CONCLUSIONS

The following conclusions could be drawn from the present work:

- By using of turbulence models and wall functions resulted in wide variation to the final prediction. Mesh fineness requires being compatible with the turbulence model/wall functions being used to obtain meaningful results.
- CFD predictions for ship resistance find good agreement with the experimental results. The work validates the use of commercial CFD software to study free surface flows. However, more exercises have to done in future to establish a better validation.
- Literature survey reveals that at present, the subject of CFD estimation of free-surface flows has not reached to high levels of maturity. In view of this global status, present results are very encouraging.

SUGGESTIONS FOR THE FUTURE WORK

- The commercial CFD software FLUENT V6.3 was utilized for analysis. The VOF technique was used for capturing the free surface and Reynolds Stress Model was adopted for turbulence closure. The mesh quality has proven to be an important issue in the computations. Future work should focus on employing a finer resolution grid than the one employed in present study. Different turbulence models can be tested in the simulations.

- Finer details of ship flow, as required by ship designers, like slamming, sea-keeping etc. can be studied in future.

NOMENCLATURE

Greek Symbols
α Volume fraction
ρ Density, kg/m^3
μ Viscosity (N-s/m^2)
Γ Transport property, m^2/s

English Symbols
F Other forces
g Acceleration due to gravity, m^2/s
n Total number of phases
p Pressure, Pa
r Raial component of velocity
t Time
u Axial component of velocity, m/s
v Velocity vector
Subscript
q qth phase

ACKNOWLEDGEMENT

I thank Dr. D.Ramesh for providing the computational resources for this work..

REFERENCES

[1.] Hirt, C. W., and Nichols, B.D., (1981) "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries," J. Comput. Phys., Vol. 39 pp. 201-225.
[2.] Fluent 6.0 Documentation, FLUENT Inc.
[3.] 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.
[4.] Patankar, S.V. (1980), Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Co., New York.