

**Universität
Rostock**



Traditio et Innovatio

Computational Methods in Fluid Dynamics

Numerical Simulation of Incompressible Flow over the Forward Facing Step

Group 5

Pawar Omkar (Matr.-Nr. 222100095)

Pawaskar Sohan (Matr.-Nr. 222200928)

Andrei Purits (Matr.-Nr. 221202242)

Shah, Meghal Devang (Matr.-Nr. 222200542)

Submitted to:

Prof. Dr.-Ing. Nikolai Kornev

Mr Muhammad Asad Yamin, M.Sc 30 January 2023

Abstract

Numerical simulations were done for incompressible flow through a forward facing step channel using $k-\omega$ SST turbulence model to study the behavior of fluid under different conditions. In this study, we have taken the Reynolds number to be 50000 and the dynamic viscosity of $3.69\text{E-}05$. No significant deviation can be seen with a change in mesh resolution and only a slight deviation can be observed between upwind and linear upwind schemes and so for small Reynolds number upwind scheme is sufficient to provide good results.

Contents

1	Introduction	4
1.1	OpenFOAM	4
1.2	Problem Description	4
2	Domain and Mesh study	5
2.1	Computational Domain	5
2.2	Mesh study	5
2.3	Viscous sublayer($y^+ < 5$)	6
2.4	Buffer Layer ($5 < y^+ < 30$)	6
2.5	Logarithmic layer ($y^+ > 30$)	6
2.6	Fine Mesh	6
2.7	Medium Mesh	7
2.8	Coarse Mesh	8
3	Boundary Conditions	9
3.1	Inlet	9
3.2	Outlet	9
3.3	Upper Wall	9
3.4	Lower Wall	9
4	Modelling and solver setup	10
4.1	Principle Governing Equations	10
4.1.1	Continuity Equation	10
4.1.2	Momentum Equation	10
4.2	Turbulence Model:k-omega SST	10
4.3	Calculation of Turbulence Kinetic Energy	11
4.3.1	Calculation of turbulence kinetic energy	11
4.3.2	Calculation of specific dissipation rate	12
4.4	Solver Setup	12
5	Residuals and convergence criteria	13
6	Results and discussion	16
6.1	Velocity Contour Graphs	16
6.2	Influence of Mesh refinement	19
6.3	Variation in discretization schemes	22
7	Conclusions	24
8	References	25

List of Figures

1	Geometry plan	5
2	Fine Mesh	7
3	Medium Mesh	7
4	Coarse Mesh	8
5	Coarse Upwind	13
6	Coarse Linear Upwind	13
7	Medium Upwind	14
8	Medium Linear Upwind	14
9	Fine Upwind	15
10	Fine Linear Upwind	15
11	Coarse Upwind velocity contour	16
12	Coarse Linear Upwind velocity contour	17
13	Medium Upwind velocity contour	17
14	Medium Linear Upwind velocity contour	18
15	Fine Upwind velocity contour	18
16	Fine Linear Upwind velocity contour	19
17	Coarse Mesh Upwind Scheme	20
18	Fine Mesh Upwind Scheme	20
19	Medium Mesh Upwind Scheme	21
20	Coarse Mesh Linear Upwind Scheme	21
21	Fine Mesh Linear Upwind Scheme	22
22	Medium Mesh Linear Upwind Scheme	22

1 Introduction

1.1 OpenFOAM

Fluid flow problems can be modeled using PDE's, the most common of which is the Navier-Stokes equation. The solution of these equations analytically is only possible for a limited number of simple geometries and flow conditions. For more complex problems, numerical methods are used to discretize the PDE's to linear algebraic equations which can be solved computationally to calculate approximate solutions. There are 3 distinct numerical techniques: finite difference, finite volume, and spectral methods. Finite difference and finite volume methods are most commonly used. They involve discretizing the spatial domain into a grid and approximating the derivatives in the PDEs using finite differences. The accuracy of a successful simulation depends on the right usage of governing equations and boundary conditions. OpenFOAM is an open-source CFD software library that provides a wide range of numerical methods for solving fluid flow and heat transfer problems. It is designed to be modular, extensible, and easy to use, and it is widely used in a variety of industries.

1.2 Problem Description

The aim of the study is to understand the Incompressible flow around a forward-facing step in a channel where only 2-Dimensional effects are considered. We simulate the fluid flow problem in OpenFOAM using the $k-\omega$ SST turbulence model with 3 different mesh resolutions and compare the results between upwind and linear upwind discretization. Finally, we need to compare the results with the given experimental data by comparing the velocity profiles from data obtained from the simulation and the given experimental data.

2 Domain and Mesh study

2.1 Computational Domain

The 2-dimensional domain consists of a forward facing step in a closed channel. Dimensions of the geometry are adapted from [1]. The step height is taken as 76mm. The inlet region upstream has a length of $5.5h = 418\text{mm}$ and the outlet region downstream has a length of $8.5h = 646\text{mm}$. To create the mesh with 3 different mesh resolutions coarse, medium, and fine block mesh utility is used. As we can see in figure 1, geometry is divided into 3 blocks for accurately defining the boundary conditions and to get the best results. Blocks 1 and 3 are the inlet zone upstream and block 2 is the outlet zone downstream.

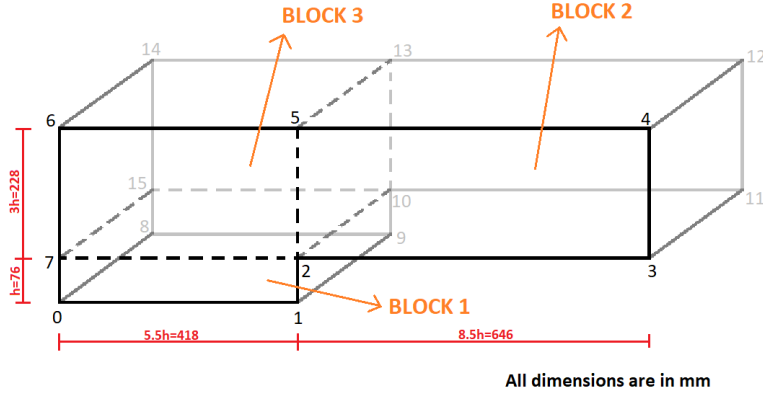


Figure 1: Geometry plan

2.2 Mesh study

It is important to have high mesh resolution close to the walls in order to capture the viscous effects of wall such as the boundary layer, which are important for turbulent flow analysis. Proper mesh refinement and modeling is necessary to ensure accurate results. The flow is divided into two regions: close to the wall (the boundary layer) and far from the wall. Near the wall, viscous forces are dominant and flow is rotational, while away from the wall inviscid and/or irrotational flow can be considered. To determine the first cell height, the calculation of y^+ is necessary. This helps distinguish different regions of flow, including the viscous sublayer, buffer layer, and logarithmic layer.

2.3 Viscous sublayer($y^+ < 5$)

The viscous sublayer is the region closest to the solid boundary, where the velocity of the fluid is affected by the viscosity of the fluid.

Mathematically it can be shown as :

$$u^+ = y^+$$

Where, u^+ is the dimensionless velocity and y^+ is the wall coordinate.

2.4 Buffer Layer ($5 < y^+ < 30$)

The buffer layer is the region immediately adjacent to the viscous sublayer. In this region, the velocity profile is still affected by the viscosity, but the turbulence intensity is beginning to increase. Neither the laminar nor the logarithmic law holds in this region, as it is a transition zone between laminar and turbulent flow.

2.5 Logarithmic layer ($y^+ > 30$)

In this region, the logarithmic law of the wall describes the relationship between the velocity profile and the distance from the wall. The logarithmic layer is a region where the velocity gradients are moderate, and the turbulence intensity is highest. The height of this layer varies according to the distance from the wall. Mathematically, it is shown as:

$$u^+ = \frac{1}{k} \ln y^+ + B$$

$$y^+ = \frac{yuT}{v}$$

Where, B and K (Karman) are constants and are obtained from measurements.

We have generated 3 types of mesh and their details are as given below:

2.6 Fine Mesh

- Block 1 contains 100 cells in x and 30 cells in y direction.
- Block 2 contains 160 cells in x and 40 cells in y direction.

- Block 3 contains 100 cells in x and 40 cells in y direction.
- Overall, 14060 elements and 28790 nodes were obtained.

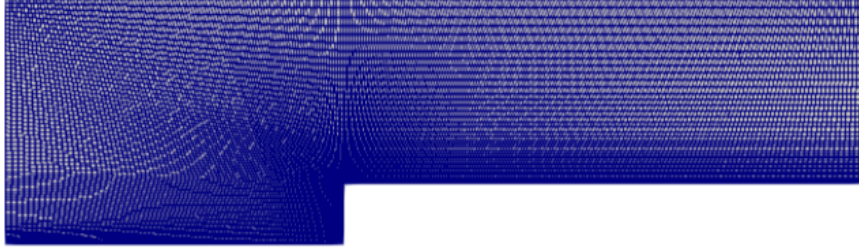


Figure 2: Fine Mesh

2.7 Medium Mesh

- Block 1 contains 70 cells in x and 21 cells in y direction.
- Block 2 contains 112 cells in x and 28 cells in y direction.
- Block 3 contains 70 cells in x and 28 cells in y direction.
- Overall, 7028 elements and 14528 nodes were obtained.

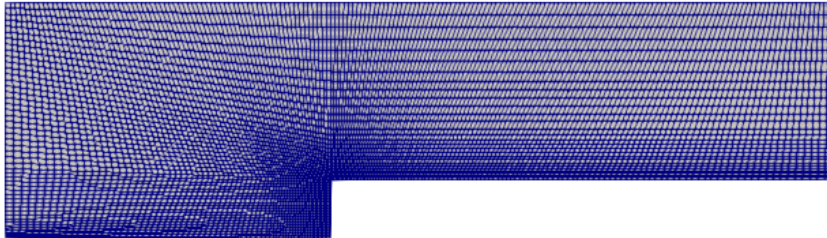


Figure 3: Medium Mesh

2.8 Coarse Mesh

- Block 1 contains 50 cells in x and 15 cells in y direction.
- Block 2 contains 80 cells in x and 20 cells in y direction.
- Block 3 contains 50 cells in x and 20 cells in y direction.
- Overall, 3680 elements and 7700 nodes were obtained.

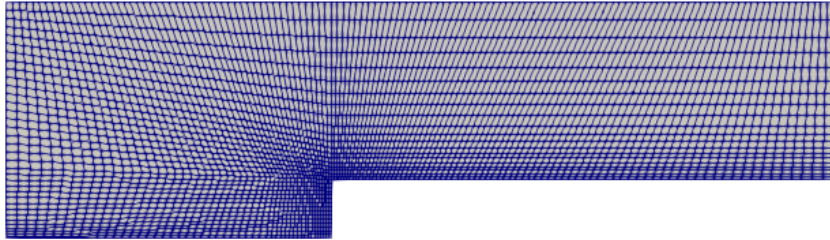


Figure 4: Coarse Mesh

3 Boundary Conditions

These are the necessary constraints that are used to define a problem. Working fluid was considered to be air and kinematic viscosity to be .Figure no. depicts the named selection in the geometry.

3.1 Inlet

- Velocity Inlet
- Uniform fixed value of in the stream-wise direction
- Zero-gradient pressure

3.2 Outlet

- Pressure Outlet
- Uniform fixed value of Zero Gauge Pressure
- Zero-Gradient Velocity

3.3 Upper Wall

- Slip boundary conditions
- Zero-Gradient Pressure

3.4 Lower Wall

- No-slip boundary condition
- Uniform fixed value of 0 m/s
- Zero-Gradient Pressure

4 Modelling and solver setup

4.1 Principle Governing Equations

4.1.1 Continuity Equation

The continuity equation states that the net rate of change of mass inside the control volume is equal to the difference between the rate at which mass flows into the control volume and the rate at which mass flows out of the control volume. As a result, Conservation of mass takes place.

$$\frac{\partial \bar{p}}{\partial t} + \frac{\partial}{\partial x_j}(\bar{p}\hat{u}_j) = 0$$

4.1.2 Momentum Equation

The momentum equation states that the rate of change of momentum is equal to the net force acting on the body. Hence, conservation of momentum takes place.

$$\frac{\partial \bar{p}\hat{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\hat{u}_j\bar{p}\hat{u}_i) + \frac{\partial p}{\partial x_i} + \frac{\partial \bar{\sigma}_{ij}}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_{ij}} = 0$$

4.2 Turbulence Model:k-omega SST

This is the best possible model which is currently in use by the industry. It is a hybrid model which combines the k-epsilon and k-omega model to take advantage of both. The k-omega model is used in the inner parts of the boundary layer and can go all the way down to the viscous sub-layer. For free-stream flows, the SST formulation transforms into k-epsilon model. This switching between turbulence models is done by a blending function. Therefore, this gives accurate predictions of the amount of flow separation under adverse pressure gradients and is also recommended for high accuracy boundary layer simulations.

The following are the transport equations for k and ω .

$$\frac{\partial k}{\partial t} + \bar{u}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}((\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j}) + P_k - \beta^* \omega k$$

$$\frac{\partial \omega}{\partial t} + \bar{u}_i \frac{\partial \omega}{\partial x_i} = \frac{\partial}{\partial x_j} ((\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j}) + \frac{\gamma}{\mu_t} P_k - \beta \omega^2 + 2(1 - F_1) \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

$$\mu_t = \frac{a_1 k}{\max(a_1 k; \omega F_2)}$$

where

F_1 blending function

$F_1 = 1$ at wall

$F_1 - > 0$ at the outer boundary of the boundary layer

4.3 Calculation of Turbulence Kinetic Energy

$$Re_H = \frac{u_0 * H}{v}$$

Consider:

$Re_H : 50000$

$H : 76mm$

$v = 3.69e10^{-5}$

$50000 = (u_0 * 76 * 10^{-3}) / (3.69 * 10^{-5})$

$u_0 = 24.27631m/s$

4.3.1 Calculation of turbulence kinetic energy

Assuming initial turbulence is isotropic.

$$k = \frac{u_x'^2 + u_y'^2 + u_z'^2}{2}$$

$$u_x'^2 = u_y'^2 = u_z'^2 = u'^2$$

$$k = \frac{3}{2} * u'^2$$

From the research paper we get Turbulence intensity $I = 0.8\%$

$k = 1.5(UI)^2$

k - Turbulent kinetic energy so, we get turbulent kinetic energy as

$k = 0.056576m^2/s^2$ (In paper it is 0.01)

4.3.2 Calculation of specific dissipation rate

Turbulence length scale is 10% of step height.

So, we get Turbulence length scale as, $L = 0.0076 \text{ m}$

$w = k^{0.5}/L$, (w = Specific dissipation rate)

So, we get specific dissipation rate as, $w = 31.29698 s^{-1}$

4.4 Solver Setup

Pressure solver was GAMG (Generalised Geometric-Algebraic Multi-grid) and the tolerance was set to 10^{-6}

The solver used for velocity (U), turbulent kinetic energy(k), Dissipation Rate(ϵ) and Specific Dissipation rate (ω) was Symmetric Gauss-Seidel smooth solver.

Scheme: linearUpwindV grad(U) and upwind

Solver: simpleFoam

5 Residuals and convergence criteria

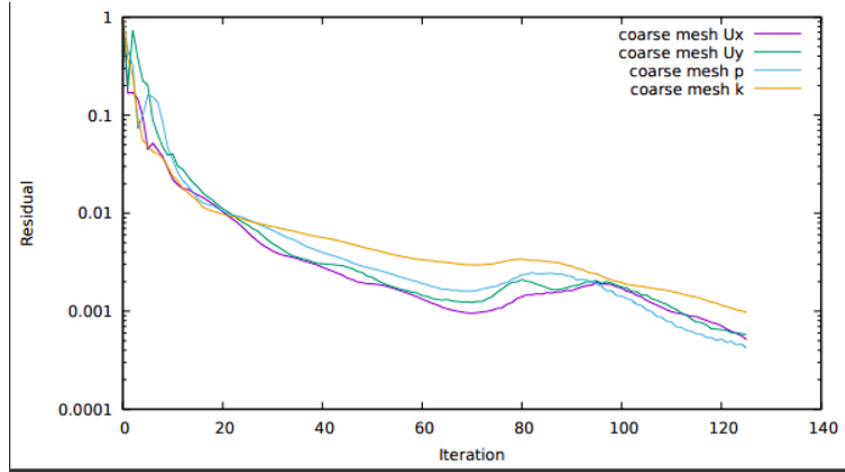


Figure 5: Coarse Upwind

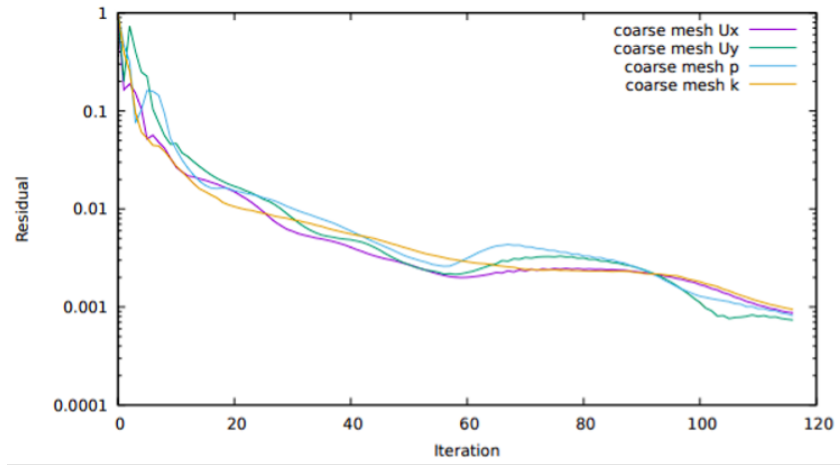


Figure 6: Coarse Linear Upwind

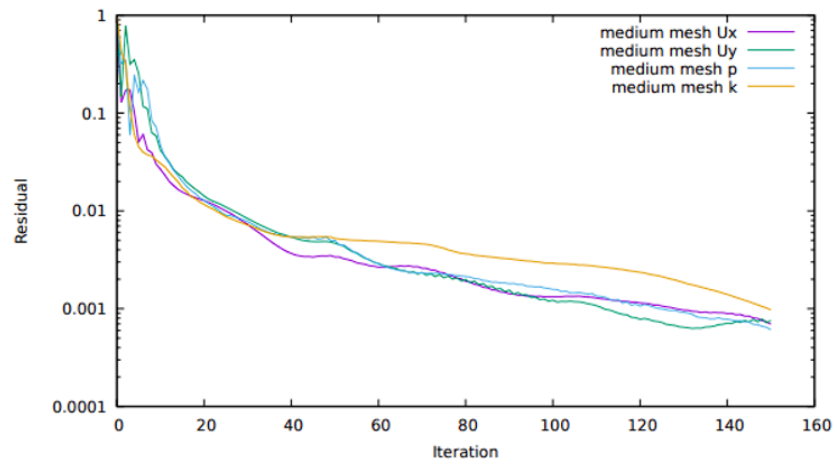


Figure 7: Medium Upwind

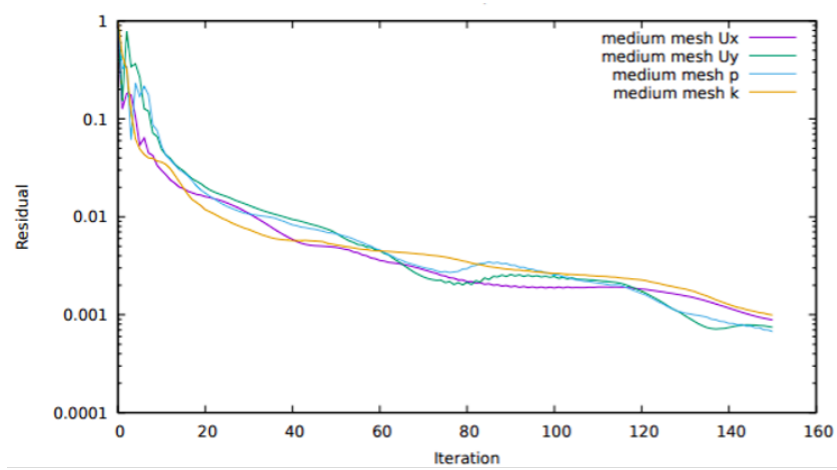


Figure 8: Medium Linear Upwind

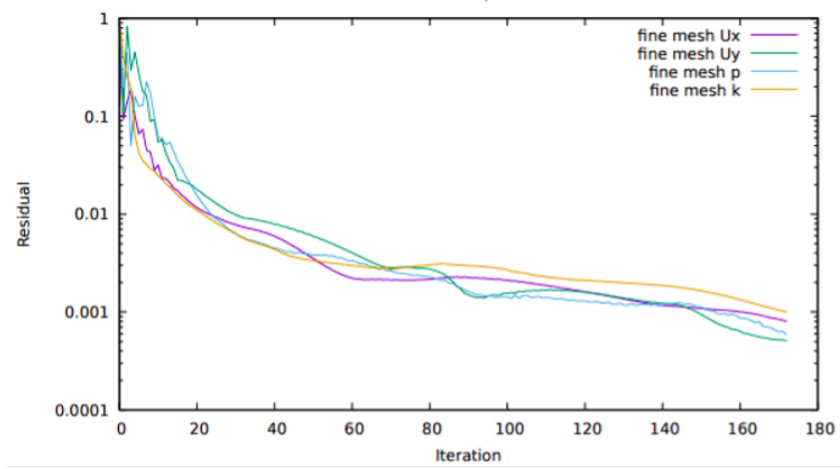


Figure 9: Fine Upwind

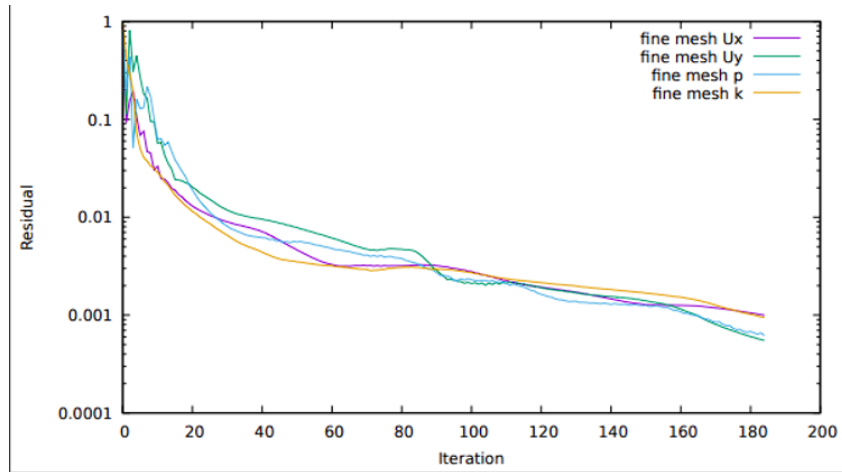


Figure 10: Fine Linear Upwind

The real time plotting was performed to check for stability. Also, the final residuals were captured and can be seen from figure. Convergence criteria was set to 10^{-6} for all field values such as x-,y-velocity and pressure field. No divergence and instability was observed throughout. Relaxation factor was 0.9 for both pressure and velocity.

6 Results and discussion

Simulated parameters, such as, the stream-wise velocity using various mesh models and discretization schemes have been evaluated and compared.

6.1 Velocity Contour Graphs

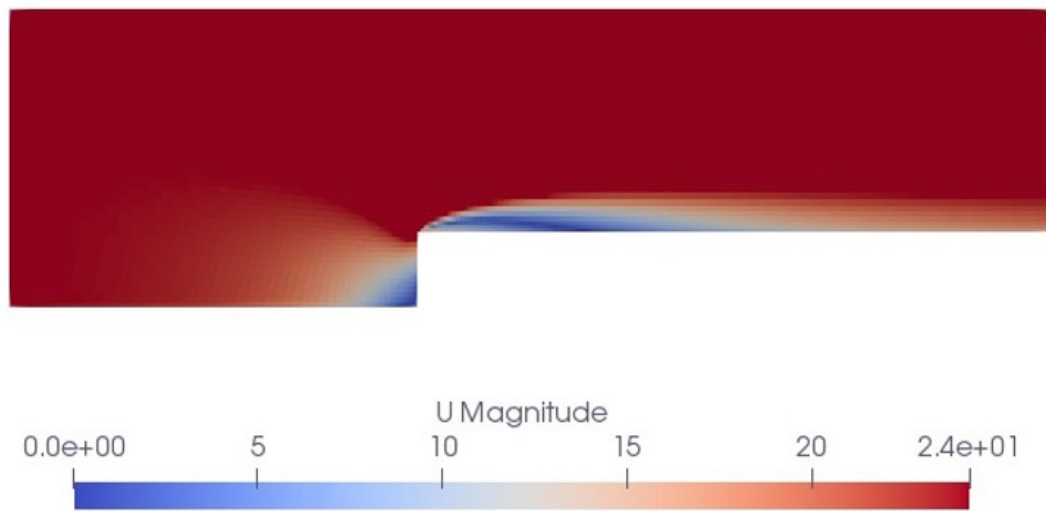


Figure 11: Coarse Upwind velocity contour

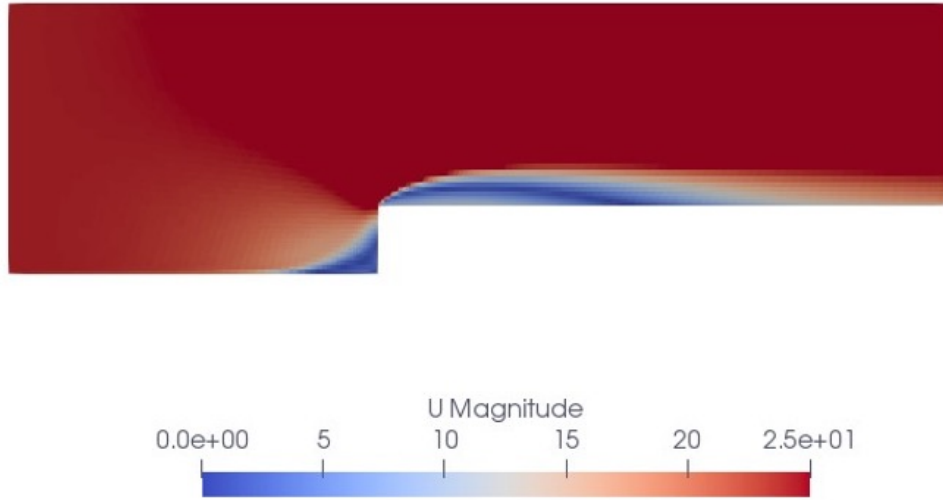


Figure 12: Coarse Linear Upwind velocity contour

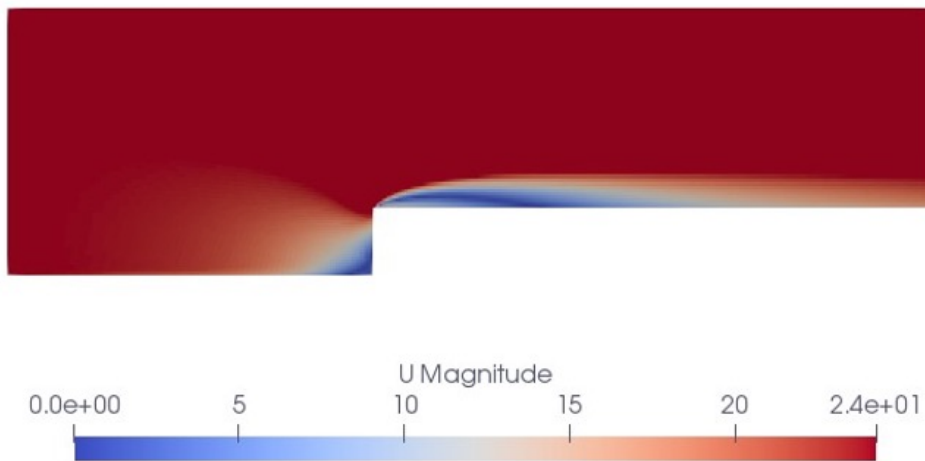


Figure 13: Medium Upwind velocity contour

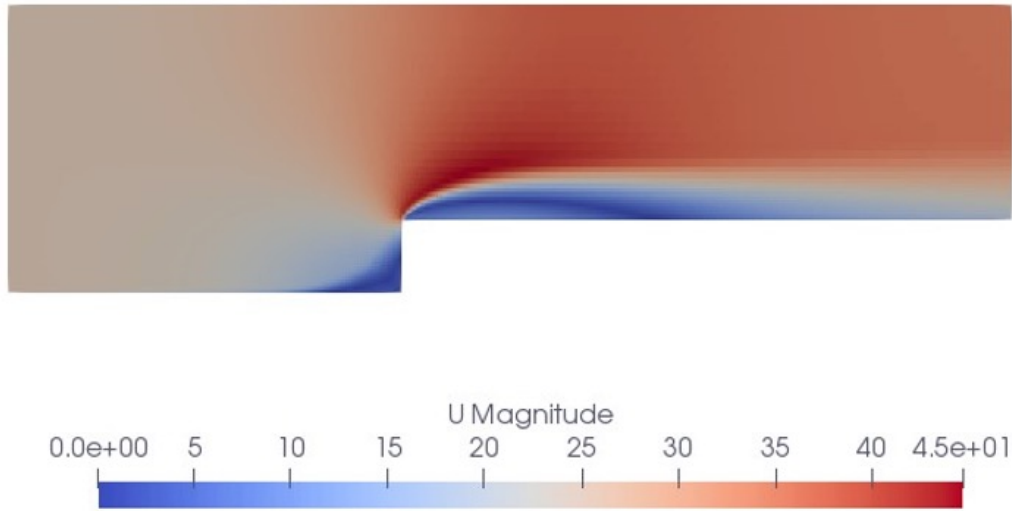


Figure 14: Medium Linear Upwind velocity contour

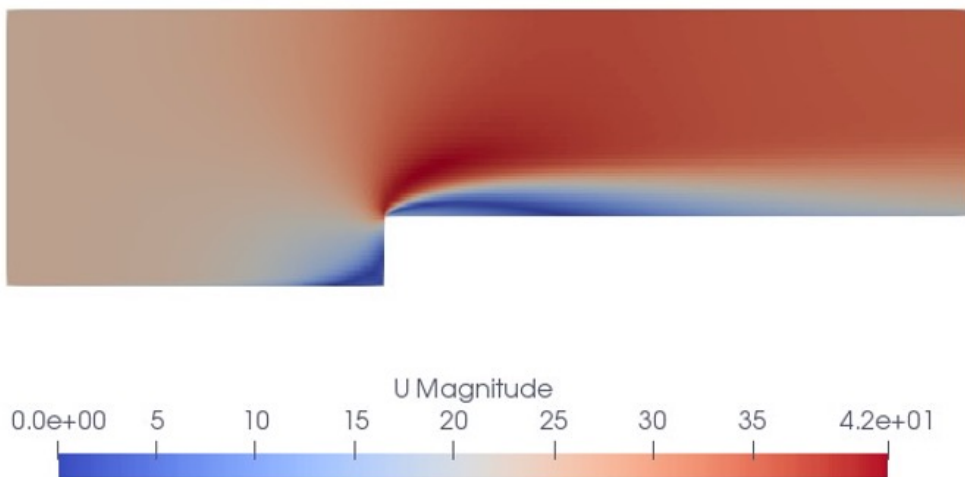


Figure 15: Fine Upwind velocity contour

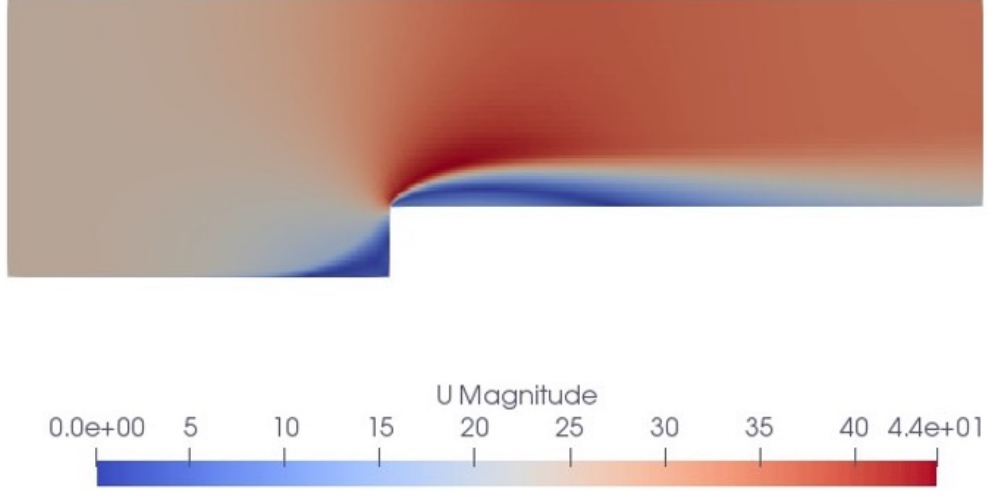


Figure 16: Fine Linear Upwind velocity contour

As seen from figure, velocity profiles are plotted and compared to (research paper no) at distinct locations to posterior step. Minimum velocity of 5 m/s, acting in the counter-flow direction, can be observed at the boundary layer creation edge. This is due to the flow separation and vortex generation that takes place in the wake region of FFS. This leads to a re-circulation zone in an important aspect in achieving flame stability in a combustion chamber. In the entire re-circulation zone, K-OmegaSST model behaves almost similarly to the experimental data, beyond which, the deviation is large and cannot be ignored.

6.2 Influence of Mesh refinement

Mesh quality and quantity plays an important role in order to obtain accuracy with optimized computational effort. No divergence was observed for the geometry containing the least number of elements. This means that the solution convergence was independent of the increment in grid refinement. This happens because hexahedral block structured grids have aspect ratio, orthogonality, skewness and other parameters within the required criterion. It can be concluded from the figure that all the meshes seem to be coinciding with each other. However a minimal deviation can be observed among them.

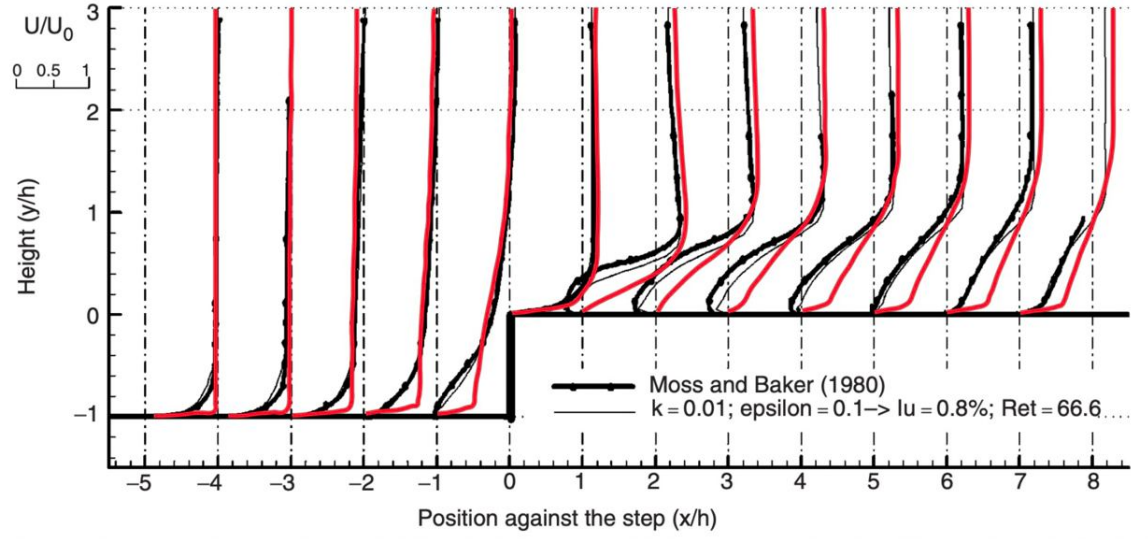


Figure 17: Coarse Mesh Upwind Scheme

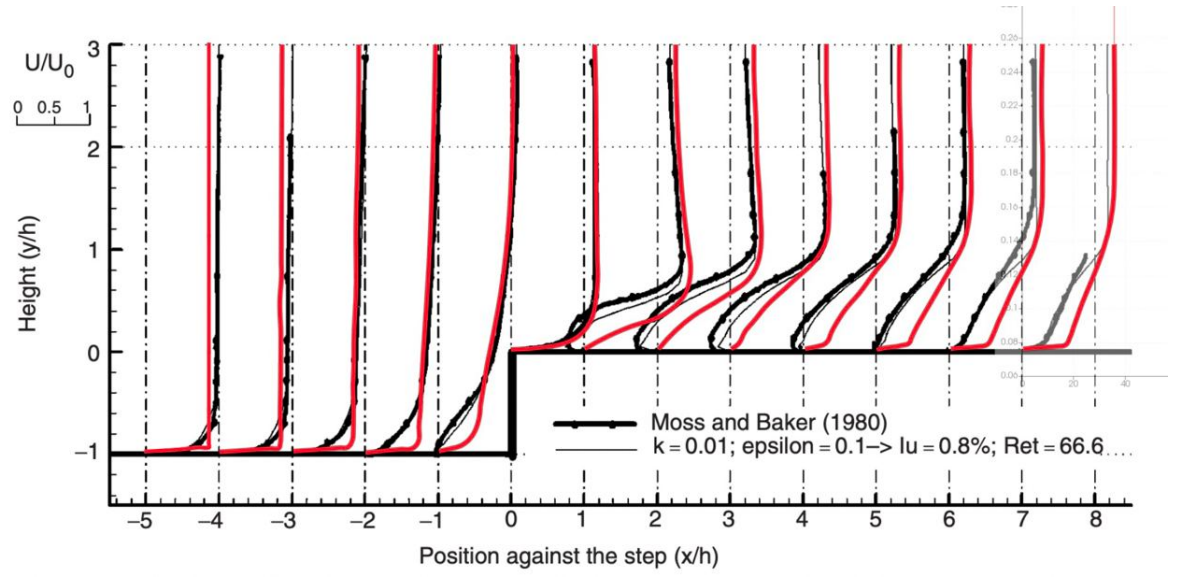


Figure 18: Fine Mesh Upwind Scheme

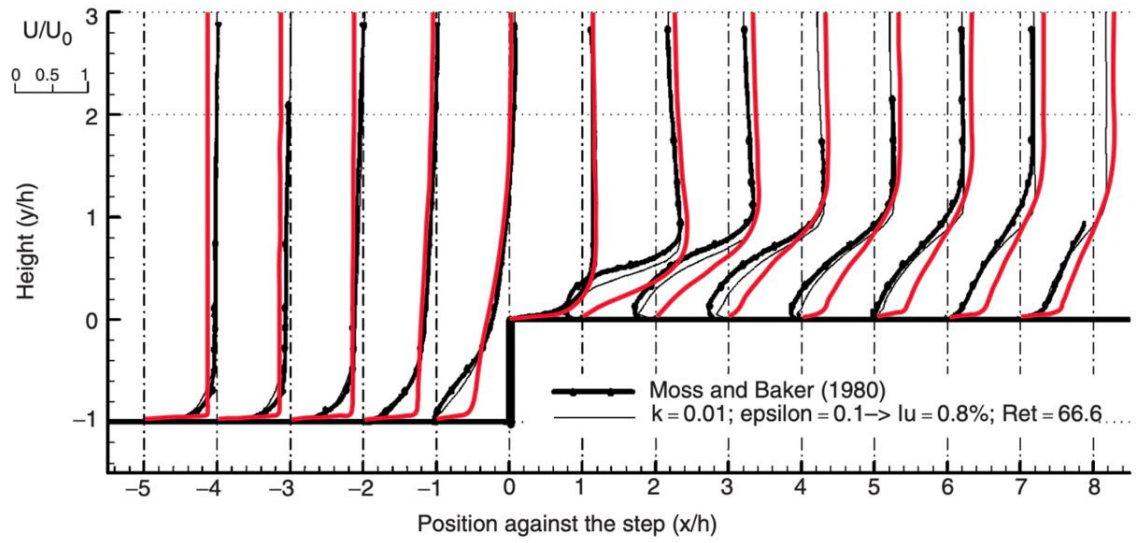


Figure 19: Medium Mesh Upwind Scheme

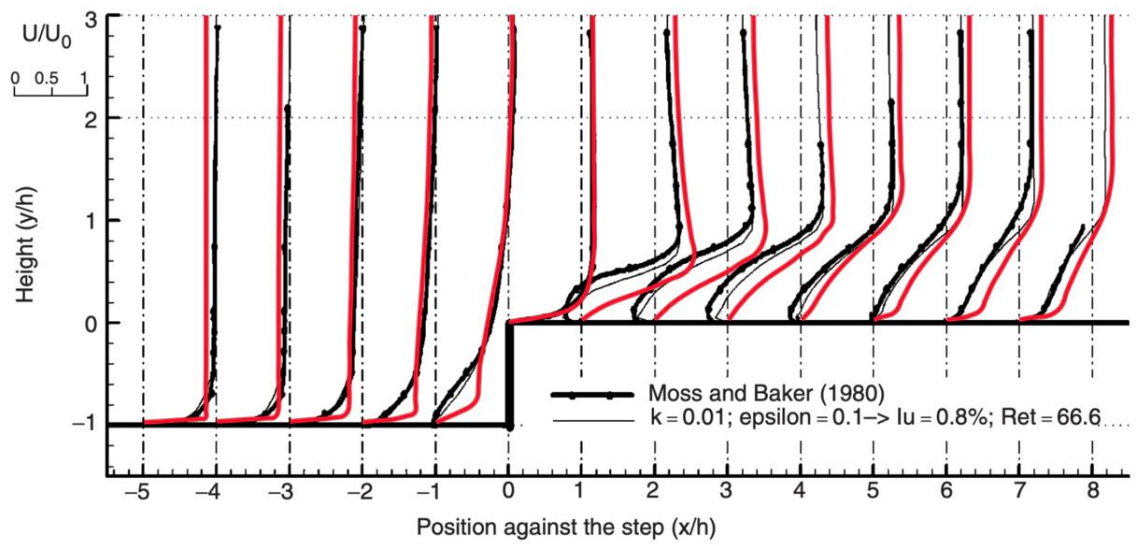


Figure 20: Coarse Mesh Linear Upwind Scheme

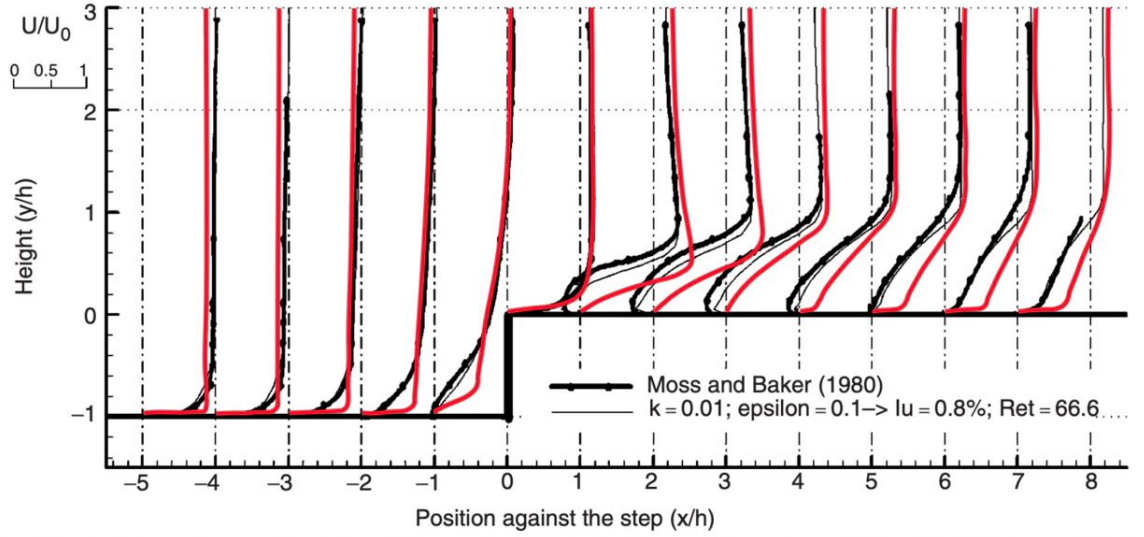


Figure 21: Fine Mesh Linear Upwind Scheme

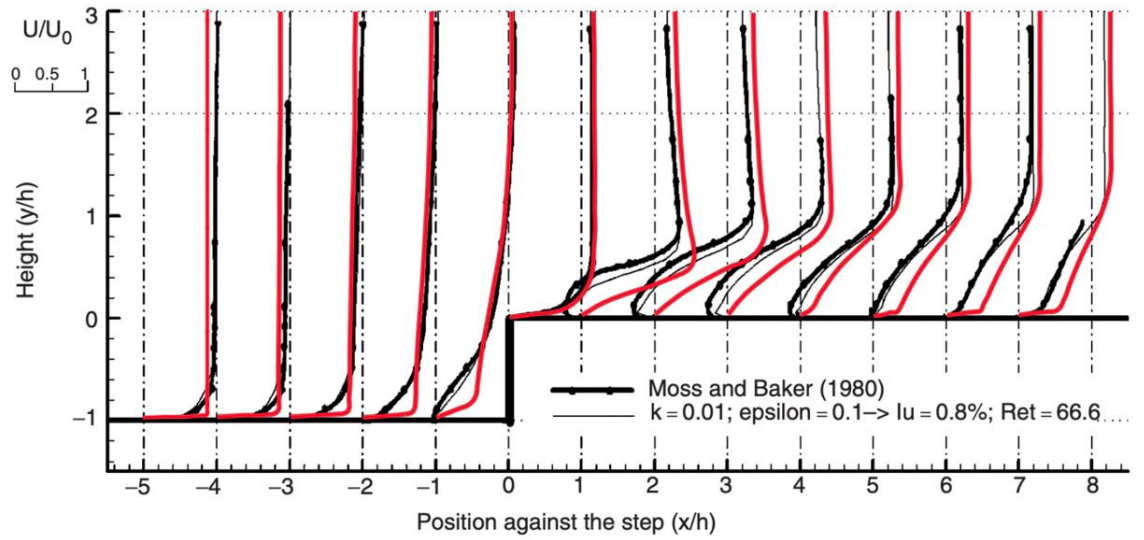


Figure 22: Medium Mesh Linear Upwind Scheme

6.3 Variation in discretization schemes

All the velocity comparison graphs of coarse, medium and fine to be inserted here.

Two discretization schemes, Upwind and Linear Upwind, were used to obtain the desired results. The velocity profiles were plotted in the wake region for both the schemes and compared to that of the experimental one. Both the schemes almost coincide throughout. However after zooming on the graph, a slight deviation among them is evident. This is due to the order of the scheme. Upwind scheme is of the first order and is slightly erroneous in capturing flow physics compared to Linear Upwind which is a second order scheme. The solution converged quicker in Upwind scheme when compared to linear Upwind.

7 Conclusions

- Delayed prediction of flow separation upstream
- Order of discretization scheme can be ignored for small Re. For instance, upwind scheme is sufficient enough to attain good accuracy
- Major source of error is modelling error that leads to difference in results
- Mesh refinement is not sufficient to reduce the inaccuracy
- To get the desired outcome, modelling errors should be reduced by considering reasonable turbulence level or utilizing a better turbulence model.

8 References

- [1] Study of Atmospheric Boundary Layer Flows Over a Coastal Cliff by Nicolas Gasset, G´erard J. Poitras, Yves Gagnon and Carl Brothers journals.sagepub.com/doi/abs/10.1260/0309524054353719
- [2] Lecture on Computational Fluid dynamics by Prof. Dr.-Ing. habil. Nikolai Kornev Prof. Dr.-Ing. habil. Irina Cherunova bookboon.com/en/lectures-on-computational-fluid-dynamics-ebook
- [3] Joel H Ferziger, Milovan Perić, and Robert L Street. Computational methods for fluid dynamics. Vol. 3. Springer, 2002