

Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows

Hrvoje Jasak

Thesis submitted for the
Degree of Doctor of Philosophy of the University of London
and
Diploma of Imperial College

Department of Mechanical Engineering
Imperial College
of Science, Technology and Medicine

June 1996

Abstract

The accuracy of numerical simulation algorithms is one of main concerns in modern Computational Fluid Dynamics. Development of new and more accurate mathematical models requires an insight into the problem of numerical errors.

In order to construct an estimate of the solution error in Finite Volume calculations, it is first necessary to examine its sources. Discretisation errors can be divided into two groups: errors caused by the discretisation of the solution domain and equation discretisation errors. The first group includes insufficient mesh resolution, mesh skewness and non-orthogonality. In the case of the second order Finite Volume method, equation discretisation errors are represented through numerical diffusion. Numerical diffusion coefficients from the discretisation of the convection term and the temporal derivative are derived. In an attempt to reduce numerical diffusion from the convection term, a new stabilised and bounded second-order differencing scheme is proposed.

Three new methods of error estimation are presented. The Direct Taylor Series Error estimate is based on the Taylor series truncation error analysis. It is set up to enable single-mesh single-run error estimation. The Moment Error estimate derives the solution error from the cell imbalance in higher moments of the solution. A suitable normalisation is used to estimate the error magnitude. The Residual Error estimate is based on the local inconsistency between face interpolation and volume integration. Extensions of the method to transient flows and the Local Residual Problem error estimate are also given.

Finally, an automatic error-controlled adaptive mesh refinement algorithm is set up in order to automatically produce a solution of pre-determined accuracy. It uses mesh refinement and unrefinement to control the local error magnitude. The method is tested on several characteristic flow situations, ranging from incompressible to supersonic flows, for both steady-state and transient problems.

Dedicated to Henry Weller

Imperial College, September 1993 - June 1996

Acknowledgements

I would like to express my sincere gratitude to my supervisors, Prof A.D. Gosman and Dr R.I. Issa for their continuous interest, support and guidance during this study.

I am also indebted to my friends and colleagues in the Prof Gosman's CFD group, particularly to Henry Weller and other people involved in the development of the FOAM C++ numerical simulation code.

The text of this Thesis has benefited from numerous valuable comments from Prof I. Demirdžić and C. Kralj.

Finally, I would like to thank Mrs N. Scott-Knight for the arrangement of many administrative matters.

The financial support provided by the Computational Dynamics Ltd. is gratefully acknowledged.

Contents

1	Introduction	41
1.1	Background	41
1.2	Previous and Related Studies	44
1.2.1	Convection Discretisation	44
1.2.2	Error Estimation	49
1.2.3	Adaptive Refinement	54
1.3	Present Contributions	58
1.4	Thesis Outline	61
2	Governing Equations	63
2.1	Governing Equations of Continuum Mechanics	63
2.2	Constitutive Relations for Newtonian Fluids	65
2.3	Turbulence Modelling	67
3	Finite Volume Discretisation	73
3.1	Introduction	73
3.2	Discretisation of the Solution Domain	75
3.3	Discretisation of the Transport Equation	77
3.3.1	Discretisation of Spatial Terms	78
3.3.1.1	Convection Term	80
3.3.1.2	Convection Differencing Scheme	81
3.3.1.3	Diffusion Term	83
3.3.1.4	Source Terms	86

3.3.2	Temporal Discretisation	87
3.3.3	Implementation of Boundary Conditions	92
3.3.3.1	Numerical Boundary Conditions	93
3.3.3.2	Physical Boundary Conditions	95
3.4	A New Convection Differencing Scheme	97
3.4.1	Accuracy and Boundedness	97
3.4.1.1	TVD Differencing Schemes	97
3.4.1.2	Convection Boundedness Criterion and the NVD Diagram	100
3.4.1.3	Convergence Problems of Flux-Limited Schemes . . .	103
3.4.2	Modification of the NVD Criterion for Unstructured Meshes .	104
3.4.3	Gamma Differencing Scheme	107
3.4.3.1	Accuracy and Convergence of the Gamma Differencing Scheme	110
3.5	Solution Techniques for Systems of Linear Algebraic Equations	111
3.6	Numerical Errors in the Discretisation Procedure	115
3.6.1	Numerical Diffusion from Convection Differencing Schemes . .	116
3.6.2	Numerical Diffusion from Temporal Discretisation	118
3.6.3	Mesh-Induced Errors	122
3.7	Numerical Examples	125
3.7.1	Numerical Diffusion from Convection Discretisation	125
3.7.2	Comparison of the Gamma Differencing Scheme with Other High-Resolution Schemes	129
3.7.2.1	Step-profile	130
3.7.2.2	$\sin^2$ -profile	130
3.7.2.3	Semi-ellipse	133
3.7.3	Numerical Diffusion from Temporal Discretisation	133
3.7.3.1	1-D Tests	135
3.7.3.2	2-D Transport of a “Bubble”	137
3.7.4	Comparison of Non Orthogonality Treatments	138

3.8	Discretisation Procedure for the Navier-Stokes System	143
3.8.1	Derivation of the Pressure Equation	145
3.8.2	Pressure-Velocity Coupling	146
3.8.2.1	The PISO Algorithm for Transient Flows	147
3.8.2.2	The SIMPLE Algorithm	148
3.8.3	Solution Procedure for the Navier-Stokes System	150
3.9	Closure	151
4	Error Estimation	153
4.1	Introduction	153
4.1.1	Error Estimators and Error Indicators	155
4.2	Requirements on an Error Estimate	157
4.3	Methods Based on Taylor Series Expansion	159
4.3.1	Richardson Extrapolation	161
4.3.2	Direct Taylor Series Error Estimate	164
4.3.3	Measuring Numerical Diffusion	167
4.4	Moment Error Estimate	168
4.4.1	Normalisation of the Moment Error Estimate	170
4.4.2	Consistency of the Moment Error Estimate	171
4.5	Residual Error Estimate	173
4.5.1	Normalisation of the Residual Error Estimate	179
4.6	Local Problem Error Estimate	181
4.6.1	Elliptic Model Problem	182
4.6.1.1	Balancing Problem in Finite Volume Method	185
4.6.2	Generalisation to the Convection-Diffusion Problem	187
4.6.3	Generalisation to the Navier-Stokes Problem	190
4.6.3.1	Error Norm for the Navier-Stokes System	191
4.6.3.2	Formulation of the Local Problem	192
4.6.4	Solution of the Local Problem	193
4.6.4.1	Solution of the Indeterminate Local Problem	194

4.6.4.2	Solution of the Determinate Local Problem	195
4.6.5	Application of the Local Problem Error Estimate	196
4.7	Error Estimation for Transient Calculations	197
4.7.1	Residual in Transient Calculations	198
4.7.2	Spatial and Temporal Error Contributions	199
4.7.3	Evolution Equation for the Error	201
4.8	Numerical Examples	202
4.8.1	Line Source in Cross-Flow	202
4.8.1.1	Mesh Aligned with the Flow	203
4.8.1.2	Non-Orthogonal Non-Aligned Mesh	208
4.8.2	Line Jet	213
4.8.3	Transient One-Dimensional Convective Transport	218
4.8.4	Local Problem Error Estimation	221
4.9	Closure	225
5	Adaptive Local Mesh Refinement and Unrefinement	227
5.1	Introduction	227
5.2	Selecting Regions of Refinement and Unrefinement	230
5.3	Mesh Refinement and Unrefinement	234
5.4	Mapping of Solution Between Meshes	237
5.5	Numerical Example	238
5.6	Closure	258
6	Case Studies	261
6.1	Introduction	261
6.2	Inviscid Supersonic Flow Over a Forward-Facing Step	263
6.3	Laminar and Turbulent Flow Over a 2-D Hill	280
6.3.1	Laminar Flow	281
6.3.2	Turbulent Flow	297
6.4	Turbulent Flow over a 3-D Swept Backward-Facing Step	329
6.5	Vortex Shedding Behind a Cylinder	355

Contents	13
6.6 Closure	365
7 Summary and Conclusions	369
7.1 Discretisation	370
7.2 Error Estimation	371
7.3 Adaptive Mesh Refinement	373
7.4 Performance of the Error-Controlled Adaptive Refinement Algorithm	374
7.5 Future Work	376
A Comparison of the Euler Implicit Discretisation and Backward Differencing in Time	379

List of Figures

3.1	Control volume.	75
3.2	Face interpolation.	81
3.3	Vectors $\mathbf{d}$ and $\mathbf{S}$ on a non-orthogonal mesh.	83
3.4	Non-orthogonality treatment in the “minimum correction” approach.	84
3.5	Non-orthogonality treatment in the “orthogonal correction” approach.	85
3.6	Non-orthogonality treatment in the “over-relaxed” approach.	85
3.7	Control volume with a boundary face.	93
3.8	Variation of ϕ around the face f	99
3.9	Sweby’s diagram.	99
3.10	Convection Boundedness Criterion in the Normalised Variable Diagram.	102
3.11	Common differencing schemes in the NVD diagram.	102
3.12	Modified definition of the boundedness criterion for unstructured meshes.	105
3.13	Shape of the profile for $0 < \tilde{\phi}_C < \beta_m$	108
3.14	Gamma differencing scheme in the NVD diagram.	110
3.15	Skewness error on the face.	124
3.16	Step-profile test setup.	126
3.17	Convection of a step-profile, $\theta = 0^\circ$, UD.	127
3.18	Convection of a step-profile, $\theta = 30^\circ$, UD.	127
3.19	Convection of a step-profile, $\theta = 30^\circ$, CD.	127
3.20	Convection of a step-profile, $\theta = 30^\circ$, UD, CD and Gamma differencing schemes.	128

3.21	Convection of a step-profile, $\theta = 45^\circ$, UD, CD and Gamma differencing schemes.	128
3.22	Convection of a step-profile, $\theta = 30^\circ$, CD, SFCD and Gamma differencing schemes.	131
3.23	Convection of a step-profile, $\theta = 30^\circ$, van Leer, SUPERBEE and Gamma differencing schemes.	131
3.24	Convection of a step-profile, $\theta = 30^\circ$, SOUCUP, SMART and Gamma differencing schemes.	131
3.25	Convection of a $\sin^2$ -profile, $\theta = 30^\circ$, CD, SFCD and Gamma differencing schemes.	132
3.26	Convection of a $\sin^2$ -profile, $\theta = 30^\circ$, van Leer, SUPERBEE and Gamma differencing schemes.	132
3.27	Convection of a $\sin^2$ -profile, $\theta = 30^\circ$, SOUCUP, SMART and Gamma differencing schemes.	132
3.28	Convection of a semi-ellipse, $\theta = 30^\circ$, CD, SFCD and Gamma differencing schemes.	134
3.29	Convection of a semi-ellipse, $\theta = 30^\circ$, van Leer, SUPERBEE and Gamma differencing schemes.	134
3.30	Convection of a semi-ellipse, $\theta = 30^\circ$, SOUCUP, SMART and Gamma differencing schemes.	134
3.31	Transport of a step-profile after 300 time-steps, four methods of temporal discretisation.	136
3.32	Transport of a half- $\sin^2$ profile after 300 time-steps, four methods of temporal discretisation.	136
3.33	Setup for the transport of the bubble.	137
3.34	Initial shape of the bubble.	139
3.35	Transport of the bubble after 800 time-steps, Euler Implicit	139
3.36	Transport of the bubble after 800 time-steps, Explicit discretisation.	139
3.37	Transport of the bubble after 800 time-steps, Crank-Nicholson.	139
3.38	Non-orthogonal test with uniform grid angle.	140

3.39	Convergence history, $\alpha_n = 10^0$	141
3.40	Convergence history, $\alpha_n = 30^0$	141
3.41	Convergence history, $\alpha_n = 40^0$	141
3.42	Convergence history, $\alpha_n = 45^0$	142
3.43	Convergence history, $\alpha_n = 65^0$	142
4.1	Hexahedral control volume aligned with the coordinate system.	163
4.2	Inconsistency between face interpolation and the integration over the cell.	174
4.3	Scaling properties of the residual error estimate.	179
4.4	Estimating the convection and diffusion transport.	180
4.5	Line source in cross-flow: mesh aligned with the flow.	203
4.6	Aligned mesh: exact solution.	204
4.7	Aligned mesh: Exact error magnitude.	204
4.8	Aligned mesh: Direct Taylor Series Error estimate.	204
4.9	Aligned mesh: Moment Error estimate.	205
4.10	Aligned mesh: Residual Error estimate.	205
4.11	Aligned mesh: scaling of the mean error.	207
4.12	Aligned mesh: scaling of the maximum error.	207
4.13	Line source in cross-flow: non-orthogonal non-aligned mesh.	208
4.14	Non-aligned mesh: exact solution.	209
4.15	Non-aligned mesh: Exact error magnitude.	209
4.16	Non-aligned mesh: Direct Taylor Series Error estimate.	209
4.17	Non-aligned mesh: Moment Error estimate.	210
4.18	Non-aligned mesh: Residual Error estimate.	210
4.19	Non-aligned mesh: scaling of the mean error.	211
4.20	Non-aligned mesh: scaling of the maximum error.	211
4.21	Non-aligned mesh: scaling of the mean error with UD.	212
4.22	Non-aligned mesh: scaling of the maximum error with UD.	212
4.23	Line jet: test setup.	214

4.24	Line jet: exact solution.	215
4.25	Line jet: exact error magnitude.	215
4.26	Line jet: Direct Taylor Series Error estimate.	215
4.27	Line jet: Moment Error estimate.	216
4.28	Line jet: Residual Error estimate.	216
4.29	Line jet: scaling of the mean error.	217
4.30	Line jet: scaling of the maximum error.	217
4.31	1-D convective transport: exact and analytical solution after 350 time-steps.	218
4.32	1-D convective transport: change in the solution during a single time-step.	219
4.33	1-D convective transport: estimated and exact single time-step error.	219
4.34	1-D convective transport: estimated and exact error after 350 time- steps.	220
4.35	Elliptic test case: Exact solution.	221
4.36	Elliptic test case: Estimated error norm distribution.	221
4.37	Line source in cross flow, aligned mesh: estimated error norm.	223
4.38	Line jet: estimated error norm.	224
5.1	Directionality of mesh refinement.	233
5.2	Refining a hexahedral cell.	235
5.3	“1-irregular” mesh.	236
5.4	Initial mesh.	240
5.5	First level of adaptive refinement.	240
5.6	Second level of adaptive refinement.	240
5.7	Third level of adaptive refinement.	240
5.8	Fourth level of adaptive refinement.	241
5.9	Fifth level of adaptive refinement.	241
5.10	Sixth level of adaptive refinement.	241
5.11	Tenth level of adaptive refinement	241

5.12	Initial distribution of the exact error.	242
5.13	Exact error distribution after two levels of adaptive refinement. . . .	243
5.14	Exact error distribution after four levels of adaptive refinement. . . .	243
5.15	Exact error distribution after six levels of adaptive refinement. . . .	243
5.16	Uniform and adaptive refinement: scaling of the mean error.	244
5.17	Uniform and adaptive refinement: scaling of the maximum error. . .	244
5.18	Uniform and adaptive refinement: scaling of the volume-weighted mean error.	245
5.19	Optimality index for uniform and adaptive refinement.	246
5.20	Refinement/unrefinement: solution at $x = 0.2m$ on the initial mesh.	248
5.21	Refinement unrefinement: solution at $x = 0.2m$ after the first level of refinement.	248
5.22	Refinement/unrefinement: solution at $x = 0.2m$ after the second level of refinement.	249
5.23	Refinement/unrefinement: solution at $x = 3m$ after the second level of refinement.	249
5.24	Refinement/unrefinement: solution at $x = 0.2m$ after the tenth level of refinement.	250
5.25	Refinement/unrefinement: solution at $x = 3m$ after the tenth level of refinement.	250
5.26	Adaptive refinement: scaling of the mean error for different error estimates and the exact error.	252
5.27	Adaptive refinement: scaling of the maximum error for different error estimates and the exact error.	252
5.28	Initial mesh for refinement unrefinement.	254
5.29	Second level of refinement unrefinement.	254
5.30	Second level of refinement-only.	254
5.31	Refinement/unrefinement: scaling of the mean error.	255
5.32	Refinement/unrefinement: scaling of the maximum error.	255

5.33	Optimality index for uniform refinement, adaptive refinement-only and adaptive refinement/unrefinement.	256
5.34	Refinement unrefinement: scaling of the mean error for different error estimates.	257
5.35	Refinement unrefinement: scaling of the maximum error for different error estimates.	257
6.1	Supersonic flow over a forward-facing step: Mach number distribution, uniform mesh, 840 CV.	264
6.2	Supersonic flow over a forward-facing step: Mach number distribution, uniform mesh, 5250 CV.	264
6.3	Supersonic flow over a forward-facing step: Mach number distribution, uniform mesh, 53000 CV.	264
6.4	Supersonic flow over a forward-facing step: error indicator Υ , uniform mesh, 840 CV.	266
6.5	Supersonic flow over a forward-facing step: error indicator Υ , uniform mesh, 5250 CV.	266
6.6	Supersonic flow over a forward-facing step: error indicator Υ , uniform mesh, 53000 CV.	266
6.7	Supersonic flow over a forward-facing step: Direct Taylor Series Error estimate for the momentum, uniform mesh, 5250 CV.	267
6.8	Supersonic flow over a forward-facing step: Moment Error estimate for the momentum, uniform mesh, 5250 CV.	267
6.9	Supersonic flow over a forward-facing step: Residual Error estimate for the momentum, uniform mesh, 5250 CV.	267
6.10	Supersonic flow over a forward-facing step coarse mesh, 840 CV. . .	268
6.11	Supersonic flow over a forward-facing step: intermediate mesh, 5250 CV.	268
6.12	Supersonic flow over a forward-facing step: first level of adaptive refinement starting from the mesh in Fig 6.10	269

6.13	Supersonic flow over a forward-facing step: second level of adaptive refinement starting from the mesh in Fig. 6.10.	269
6.14	Supersonic flow over a forward-facing step: third level of adaptive refinement starting from the mesh in Fig. 6.10.	269
6.15	Detail of the adaptively refined mesh.	270
6.16	Supersonic flow over a forward-facing step: Mach number distribution on the adaptively refined mesh after three levels of refinement. .	271
6.17	Supersonic flow over a forward-facing step: Mach number distribution on the adaptively refined mesh after five levels of refinement. .	271
6.18	Supersonic flow over a forward-facing step: third level of adaptive refinement starting from the mesh in Fig. 6.11.	272
6.19	Supersonic flow over a forward-facing step: first level of adaptive refinement/unrefinement starting from the mesh in Fig. 6.11. . . .	273
6.20	Supersonic flow over a forward-facing step: second level of adaptive refinement/unrefinement starting from the mesh in Fig. 6.11. . . .	273
6.21	Supersonic flow over a forward-facing step: third level of adaptive refinement/unrefinement starting from the mesh in Fig. 6.11. . . .	273
6.22	Supersonic flow over a forward-facing step: Mach number distribution for the mesh in Fig. 6.18.	274
6.23	Supersonic flow over a forward-facing step: Mach number distribution for the mesh in Fig. 6.21.	274
6.24	Supersonic flow over a forward-facing step: scaling of the mean error for uniform refinement.	277
6.25	Supersonic flow over a forward-facing step: scaling of the maximum error for uniform refinement.	277
6.26	Supersonic flow over a forward-facing step: scaling of the mean error for adaptive refinement.	278
6.27	Supersonic flow over a forward-facing step: scaling of the maximum error for adaptive refinement.	278

6.28	Supersonic flow over a forward-facing step: scaling of the mean error for refinement unrefinement.	279
6.29	Supersonic flow over a forward-facing step: scaling of the maximum error for refinement/unrefinement.	279
6.30	Two-dimensional hill test case.	280
6.31	Two-dimensional hill: uniform mesh with 2044 CV.	281
6.32	Laminar flow over a 2-D hill: velocity field.	282
6.33	Laminar flow over a 2-D hill: pressure field.	282
6.34	Laminar flow over a 2-D hill: vector velocity error, uniform mesh, 2044 CV.	283
6.35	Laminar flow over a 2-D hill: velocity error magnitude, uniform mesh, 2044 CV.	283
6.36	Laminar flow over a 2-D hill: velocity error magnitude, uniform mesh, 8584 CV.	284
6.37	Laminar flow over a 2-D hill: Direct Taylor Series Error estimate for the velocity, uniform mesh, 2044 CV.	285
6.38	Laminar flow over a 2-D hill: Moment Error estimate for the velocity, uniform mesh, 2044 CV.	285
6.39	Laminar flow over a 2-D hill: Residual Error estimate for the velocity, uniform mesh, 2044 CV.	285
6.40	Laminar flow over a 2-D hill: scaling of the mean error for uniform refinement.	286
6.41	Laminar flow over a 2-D hill: scaling of the maximum error for uniform refinement.	286
6.42	Laminar flow over a 2-D hill. estimated error norm, uniform mesh, 2044 CV.	287
6.43	Laminar flow over a 2-D hill: scaling of the relative error in dissipation for uniform refinement.	288
6.44	Laminar flow over a 2-D hill. mesh after the first level of refinement.	289
6.45	Laminar flow over a 2-D hill: mesh after the second level of refinement.	289

6.46	Laminar flow over a 2-D hill: mesh after the third level of refinement.	289
6.47	Laminar flow over a 2-D hill: first level of refinement/unrefinement. .	290
6.48	Laminar flow over a 2-D hill: second level of refinement/unrefinement.	290
6.49	Laminar flow over a 2-D hill: Residual Error estimate after the first level of refinement.	292
6.50	Laminar flow over a 2-D hill: Residual Error estimate after the sec- ond level of refinement.	292
6.51	Laminar flow over a 2-D hill: Residual Error estimate after the third level of refinement.	292
6.52	Laminar flow over a 2-D hill: Residual Error estimate on the uniform mesh, 8584 CV-s.	293
6.53	Laminar flow over a 2-D hill: Residual Error estimate after the first level of refinement/unrefinement from the fine mesh.	293
6.54	Laminar flow over a 2-D hill: Residual Error estimate after the sec- ond level of refinement/unrefinement from the fine mesh.	293
6.55	Laminar flow over a 2-D hill: second level of refinement-only from the fine mesh.	294
6.56	Laminar flow over a 2-D hill: Residual Error estimate after the sec- ond level of refinement/unrefinement.	294
6.57	Laminar flow over a 2-D hill: scaling of the maximum Moment Error for different types of refinement.	295
6.58	Laminar flow over a 2-D hill: scaling of the maximum Residual Error for different types of refinement.	295
6.59	Turbulent flow over a 2-D hill: velocity field, uniform mesh, 2044 CV.	299
6.60	Turbulent flow over a 2-D hill: turbulent kinetic energy field, uniform mesh, 2044 CV.	299
6.61	Turbulent flow over a 2-D hill: dissipation of turbulent kinetic energy, uniform mesh, 2044 CV.	299
6.62	Turbulent flow over a 2-D hill: Direct Taylor Series Error estimate for velocity, uniform mesh, 2044 CV.	300

6.63	Turbulent flow over a 2-D hill: Moment Error estimate for velocity, uniform mesh, 2044 CV.	300
6.64	Turbulent flow over a 2-D hill: Residual Error estimate for velocity, uniform mesh, 2044 CV.	300
6.65	Turbulent flow over a 2-D hill: Direct Taylor Series Error estimate for k , uniform mesh, 2044 CV.	301
6.66	Turbulent flow over a 2-D hill: Moment Error estimate for k , uniform mesh, 2044 CV.	301
6.67	Turbulent flow over a 2-D hill: Residual Error estimate for k , uniform mesh, 2044 CV.	301
6.68	Turbulent flow over a 2-D hill: Direct Taylor Series Error estimate for ϵ , uniform mesh, 2044 CV.	302
6.69	flow over a 2-D hill: Moment Error estimate for ϵ , uniform mesh, 2044 CV.	302
6.70	Turbulent flow over a 2-D hill: Residual Error estimate for ϵ , uniform mesh, 2044 CV.	302
6.71	Turbulent flow over a 2-D hill: estimated error norm for velocity, uniform mesh, 2044 CV.	305
6.72	Turbulent flow over a 2-D hill: estimated error norm for k , uniform mesh, 2044 CV.	305
6.73	Turbulent flow over a 2-D hill: estimated error norm for ϵ , uniform mesh, 2044 CV.	305
6.74	Turbulent flow over a 2-D hill: scaling of the mean velocity error for uniform refinement.	306
6.75	Turbulent flow over a 2-D hill: scaling of the maximum velocity error for uniform refinement.	306
6.76	Turbulent flow over a 2-D hill. scaling of the mean error in k for uniform refinement.	307
6.77	Turbulent flow over a 2-D hill: scaling of the maximum error in k for uniform refinement	307

6.78	Turbulent flow over a 2-D hill: scaling of the mean error in ϵ for uniform refinement.	308
6.79	Turbulent flow over a 2-D hill: scaling of the maximum error in ϵ for uniform refinement.	308
6.80	Turbulent flow over a 2-D hill: mesh after the first level of refinement.	311
6.81	Turbulent flow over a 2-D hill: mesh after the second level of refinement.	311
6.82	Turbulent flow over a 2-D hill: mesh after the third level of refinement.	311
6.83	Turbulent flow over a 2-D hill: Moment Error estimate for velocity after the first level of adaptive refinement.	312
6.84	Turbulent flow over a 2-D hill: Moment Error estimate for velocity after the second level of adaptive refinement.	312
6.85	Turbulent flow over a 2-D hill: Moment Error for velocity estimate after the third level of adaptive refinement.	312
6.86	Turbulent flow over a 2-D hill: Residual Error estimate after the first level of adaptive refinement.	313
6.87	Turbulent flow over a 2-D hill: Residual Error estimate after the second level of adaptive refinement.	313
6.88	Turbulent flow over a 2-D hill: Residual Error estimate after the third level of adaptive refinement.	313
6.89	Turbulent flow over a 2-D hill: second level of refinement-only from the fine mesh.	314
6.90	Turbulent flow over a 2-D hill: second level of refinement/unrefinement from the fine mesh.	314
6.91	Turbulent flow over a 2-D hill: Moment Error estimate for velocity after two levels of refinement/unrefinement.	316
6.92	Turbulent flow over a 2-D hill: Residual Error estimate for k after two levels of refinement/unrefinement.	316
6.93	Turbulent flow over a 2-D hill: scaling of the maximum Moment Error for velocity for different types of refinement.	318

6.94	Turbulent flow over a 2-D hill: scaling of the maximum Moment Error for velocity without the near-wall cells for different types of refinement.	318
6.95	Turbulent flow over a 2-D hill: scaling of the maximum Moment Error for k for different types of refinement.	319
6.96	Turbulent flow over a 2-D hill: scaling of the maximum Residual Error for k for different types of refinement.	319
6.97	Turbulent flow over a 2-D hill: velocity distribution at $x = 0$ for uniform meshes.	320
6.98	Turbulent flow over a 2-D hill: velocity distribution at $x = 0$, first level of refinement.	320
6.99	Turbulent flow over a 2-D hill: velocity distribution at $x = 0$, second level of refinement.	320
6.100	Turbulent flow over a 2-D hill: velocity distribution at $x = 134$ for uniform meshes.	321
6.101	Turbulent flow over a 2-D hill: velocity distribution at $x = 134$, first level of refinement.	321
6.102	Turbulent flow over a 2-D hill: velocity distribution at $x = 134$, second level of refinement.	321
6.103	Turbulent flow over a 2-D hill: distribution of k at $x = 0$ for uniform meshes.	322
6.104	Turbulent flow over a 2-D hill: distribution of k at $x = 0$, first level of refinement.	322
6.105	Turbulent flow over a 2-D hill: distribution of k at $x = 0$, second level of refinement.	322
6.106	Turbulent flow over a 2-D hill: distribution of k at $x = 134$ for uniform meshes.	323
6.107	Turbulent flow over a 2-D hill: distribution of k at $x = 134$ first level of refinement.	323

6.108 Turbulent flow over a 2-D hill: distribution of k at $x = 134$, second level of refinement.	323
6.109 Turbulent flow over a 2-D hill: uniform mesh for the low- Re calculation.	325
6.110 Turbulent flow over a 2-D hill: velocity field for the low- Re turbulence model.	325
6.111 Turbulent flow over a 2-D hill: maximum error reduction for velocity with adaptive refinement and the low- Re turbulence model.	326
6.112 Turbulent flow over a 2-D hill: maximum error reduction for k with adaptive refinement and the low- Re turbulence model.	328
6.113 Turbulent flow over a 2-D hill: maximum error reduction for ϵ with adaptive refinement and the low- Re turbulence model.	328
6.114 3-D swept backward facing step: test setup.	329
6.115 3-D swept backward-facing step: coarse uniform mesh, 16625 CV. . .	331
6.116 3-D swept backward-facing step: stream ribbons close to the inlet. .	333
6.117 3-D swept backward-facing step: stream ribbon in the vortex. . . .	333
6.118 3-D swept backward-facing step: velocity field cut, $y/H = 0.125$. . .	334
6.119 3-D swept backward-facing step: velocity field cut, $y/H = 0.5$	334
6.120 3-D swept backward-facing step: velocity field cut, $y/H = 1.125$. . .	335
6.121 3-D swept backward-facing step: surface pressure.	335
6.122 3-D swept backward-facing step: near-surface k distribution.	336
6.123 3-D swept backward-facing step: k iso-surface.	336
6.124 3-D swept backward-facing step: Moment Error estimate for velocity.	338
6.125 3-D swept backward-facing step: Residual Error estimate for velocity.	338
6.126 3-D swept backward-facing step: Moment Error estimate for k	339
6.127 3-D swept backward-facing step: Residual Error estimate for k	339
6.128 3-D swept backward-facing step: Moment Error estimate for ϵ	340
6.129 3-D swept backward-facing step: Residual Error estimate for ϵ	340
6.130 3-D swept backward-facing step: coarse uniform mesh at the corner of the triangular part.	343

6.131 3-D swept backward-facing step: mesh detail after the first level of refinement.	343
6.132 3-D swept backward-facing step: mesh detail after the second level of refinement.	344
6.133 3-D swept backward-facing step: mesh detail after the third level of refinement.	344
6.134 3-D swept backward-facing step: iso-surface of the Moment Error estimate for velocity on the initial mesh.	346
6.135 3-D swept backward-facing step: iso-surface of the Moment Error estimate for velocity after the first level of refinement.	346
6.136 3-D swept backward-facing step: iso-surface of the Moment Error estimate for velocity after the second level of refinement.	347
6.137 3-D swept backward-facing step: iso-surface of the Moment Error estimate for velocity after the third level of refinement.	347
6.138 3-D swept backward-facing step: iso-surface of the Residual Error estimate for k on the initial mesh.	348
6.139 3-D swept backward-facing step: iso-surface of the Residual Error estimate for k after the first level of refinement.	348
6.140 3-D swept backward-facing step: iso-surface of the Residual Error estimate for k after the second level of refinement.	349
6.141 3-D swept backward-facing step: iso-surface of the Residual Error estimate for k after the third level of refinement.	349
6.142 3-D swept backward-facing step: scaling of the maximum velocity error for adaptive refinement.	350
6.143 3-D swept backward-facing step: scaling of the mean velocity error for adaptive refinement.	350
6.144 3-D swept backward-facing step: scaling of the maximum velocity error for uniform and adaptive refinement.	351
6.145 3-D swept backward-facing step: scaling of the maximum k error for uniform and adaptive refinement.	352

6.146 3-D swept backward-facing step: scaling of the maximum k error for adaptive refinement.	352
6.147 3-D swept backward-facing step: scaling of the maximum ϵ error for adaptive refinement.	353
6.148 Vortex shedding behind a cylinder: test setup.	355
6.149 Vortex shedding: uniform mesh.	355
6.150 Vortex shedding: velocity field.	356
6.151 Vortex shedding: pressure field.	356
6.152 Vortex shedding: enstrophy distribution.	356
6.153 Vortex shedding: pressure trace for different methods of temporal discretisation.	357
6.154 Vortex shedding: full Residual Error for EI on $Co = 0.4$	358
6.155 Vortex shedding: spatial Residual Error for EI on $Co = 0.4$	358
6.156 Vortex shedding: temporal Residual Error for EI on $Co = 0.4$	360
6.157 Vortex shedding: temporal Residual Error for BD on $Co = 0.4$	360
6.158 Vortex shedding: temporal Residual Error for CN on $Co = 2$	360
6.159 Vortex shedding: full Residual Error for EI on $Co = 2$	361
6.160 Vortex shedding: temporal Residual Error for EI on $Co = 2$	361
6.161 Vortex shedding: pressure trace for the Euler Implicit discretisation on two Courant numbers.	362
6.162 Vortex shedding: adaptively refined mesh changing in time.	363
6.163 Vortex shedding: mesh refinement based on the error in the complete shedding cycle.	363
6.164 Vortex shedding: spatial Residual Error after the first level of refinement.	364
6.165 Vortex shedding: spatial Residual Error after the second level of refinement.	364
6.166 Vortex shedding: temporal Residual Error after the second level of refinement.	364

List of Tables

4.1	Determinate local problem: accuracy of the approximate solution. . .	222
4.2	Indeterminate local problem: accuracy of the approximate solution. .	223
4.3	Line source in cross flow, aligned mesh: global error norm and effectivity index.	224
4.4	Line jet: global error norm and effectivity index.	224
6.1	Supersonic flow over a forward-facing step: number of cells for adaptive and uniform refinement starting from the coarse mesh.	275
6.2	Supersonic flow over a forward-facing step: number of cells for refinement-only and refinement/unrefinement starting from the intermediate mesh.	275
6.3	The profile of the hill.	280
6.4	3-D swept backward-facing step: iso-surface level for the Moment and Residual Error estimates.	337
6.5	3-D swept backward-facing step: number of cells for adaptive and uniform refinement starting from the coarse mesh.	345
6.6	3-D swept backward-facing step: number of cells for refinement-only and refinement/unrefinement starting from the fine mesh.	354

Nomenclature

Latin Characters

- $1, 2, 3$ principal vectors of inertia
- $\mathbf{a}$ general vector property
- a_N matrix coefficient corresponding to the neighbour N
- a_P central coefficient
- Co Courant number
- D_C – convection part of the temporal error
- D_D diffusion part of the temporal error
- D_S source term part of the temporal error
- $\mathbf{d}$ vector between P and N
- $\mathbf{d}_n$ – vector between the cell centre and the boundary face
- E exact error
- E_0 – desired error level
- E_c numerical diffusion from convection
- E_t numerical diffusion from temporal discretisation
- E_d numerical diffusion from mesh non-orthogonality
- E_s numerical diffusion from mesh skewness
- e total specific energy, solution error

e_m	Moment Error estimate
e_r	Residual Error estimate
e_t	Taylor Series Error estimate
e_{num}	numerical diffusion error
e_M	kinetic energy
F	mass flux through the face
F_{con}	convection transport coefficient
F_{diff}	diffusion transport coefficient
F_{norm}	normalisation factor for the residual
f	face. point in the centre of the face
f^+	downstream face
f^-	upstream face
f_i	point of interpolation on the face
f_x	interpolation factor
$\mathbf{g}$	body force
g_b	boundary condition on the fixed gradient boundary
H	transport part
h	mesh size
$\mathbf{I}$	unit tensor
$\mathbf{i}, \mathbf{j}$	unit vectors
$\mathbf{k}$	non-orthogonal part of the face area vector
k	turbulent kinetic energy
l	desired local mesh size
$\mathbf{M}$	second geometric moment tensor
$\mathbf{m}$	skewness correction vector

m_a	second moment of $\mathbf{a}$
m_ϕ	second moment of ϕ
$\mathcal{N}$	number of identically performed experiments
N	point in the centre of the neighbouring control volume
P	pressure, point in the centre of the control volume
$\mathbf{p}$	position difference vector
p	kinematic pressure, order of accuracy
Q	volume energy source
Q_S	surface source
Q_V	volume source
$\mathbf{q}$	heat flux
R_P	right-hand size of the algebraic equation
r	smoothness monitor for TVD differencing schemes
Re	Reynolds number
res_m	moment imbalance
res_F	transient residual
res_L	spatial residual
res_P	cell residual
res_T	temporal residual
$\mathbf{S}$	outward-pointing face area vector
$\mathbf{S}_f$	face area vector
S_ϕ	source term
S_e	error source term
S_p	linear part of the source term
S_u	constant part of the source term

s	specific entropy
T	temperature, time-scale
t	time
t_{ref}	time-step indicator
$\mathbf{U}$	velocity
U_{trans}	effective transport velocity
u	specific internal energy
V	volume
V_M	material volume
V_P	volume of the cell
$\mathbf{x}$	position vector

Greek Characters

α	under-relaxation factor
α	non-orthogonality angle
α_p	pressure under-relaxation factor
α_U	velocity under-relaxation factor
β_m	parameter of the Gamma differencing scheme
Γ_D	numerical diffusion tensor from mesh non-orthogonality
Γ_N	numerical diffusion tensor from convection discretisation
Γ_{num}	numerical diffusion tensor
Γ_S	numerical diffusion tensor from mesh skewness
Γ_T	numerical diffusion tensor from temporal discretisation
Γ_ϕ	diffusivity
γ	blending factor, heat capacity ratio

Δ	orthogonal part of the face area vector
ϵ	dissipation rate of turbulent kinetic energy
ζ	– effectivity index
θ	mesh-to-flow angle
λ	heat conductivity
λ_{ref}	refinement criterion
λ_{unref}	unrefinement criterion
μ	dynamic viscosity
ν	– kinematic viscosity
ν_t	kinematic eddy viscosity
ξ	directionality parameter
ρ	density
σ	stress tensor
τ	error directionality
χ	general tensorial property
Ψ	TVD limiter
Φ	exact solution
ϕ	general scalar property

Superscripts

$\mathbf{q}^T$	transpose
$\bar{q}$	mean
q'	fluctuation around the mean value
q^n	new time-level
q^o	old time-level

q^{oo} “second old” time-level

$\mathbf{q}$ unit vector

$\tilde{q}$ normalised

Subscripts

q_f value on the face

q_b value on the boundary face

Abbreviations

BD	Blended Differencing
Bi-CG	Bi-Conjugate Gradient
BSUD	Bounded Streamwise Upwind Differencing
CAD	Computer-Aided Design
CAE	Computer-Aided Engineering
CAM	Computer-Aided Manufacturing
CBC	Convection Boundedness Criterion
CD	Central Differencing
CFD	Computational Fluid Dynamics
CG	Conjugate Gradient
CV	Control Volume
DNS	Direct Numerical Simulation
EOC	Experimental Order of Convergence
FCT	Flux-Corrected Transport
FV	Finite Volume
FVM	Finite Volume Method
ICCG	Incomplete Cholesky Conjugate Gradient
LDA	Laser-Doppler Anemometry
LES	Large-Eddy Simulation
LOADS	Locally Analytical Differencing Scheme

LUD	Linear Upwind Differencing
NDR	Numerical Diffusion Ratio
NVA	Normalised Variable Approach
NVD	Normalised Variable Diagram
NURBS	Non-Uniform Rational b -Spline
PISO	Pressure-Implicit with Splitting of Operators
QUICK	– Quadratic Upstream Interpolation for Convective Kinematics
RNG	Renormalisation Group
SFCD	Self-Filtered Central Differencing
SHARP	Simple High-Accuracy Resolution Program
SIMPLE	– Semi-Implicit Method for Pressure-Linked Equations
SMART	Sharp and Monotonic Algorithm for Realistic Transport
SOUCUP	Second-Order Upwind-Central Differencing-First-Order-Upwind
TV	– Total Variation
TVD	Total Variation Diminishing
UD	Upwind Differencing
UMIST	Upstream Monotonic Interpolation for Scalar Transport

Chapter 1

Introduction

1.1 Background

Numerical tools for structural analysis have been widely accepted in the modern engineering community. The concept of Computer-Aided Design (CAD), Computer-Aided Manufacturing (CAM) and more generally, Computer-Aided Engineering (CAE) provides the possibility of optimising the design of the final product in many different ways. Quick and accurate structural analysis is an important part of the development process and numerical structures analysis packages are integrated into most modern CAD systems.

The performance of many products, ranging from kitchen appliances to nuclear submarines, depends not only on their structural properties, but also on the characteristics of heat transfer, fluid flow and even fluid-solid interaction which play an important role in their functionality. In order to improve their design, it is necessary to extend the optimisation by including the fluid flow phenomena into the numerical simulation. The progress in this area has been much slower – the flow problems generally require a solution of the systems of coupled non-linear partial differential equations, which are more difficult to solve.

Computational Fluid Dynamics (CFD) provides the methods for numerical simulation of fluid flows. In spite of the fact that CFD analysis is regularly done in some areas of engineering, it is still not a widely accepted design tool. The complexity of

flow regimes in, for example, internal combustion engines, is such that an accurate and predictive simulation becomes very expensive in terms of time and computer resources. In order to simulate the features of the flow well, complicated models and accurate solutions are needed.

The accuracy of numerical solutions represents an interesting field. A numerical solution is obtained following a set of rules that provide a discrete description of the governing equations and the solution domain. Its accuracy is determined from the correspondence between the exact solution and its numerical approximation. The judgement on the solution accuracy should therefore be done by comparing it with the exact solution, which is usually unavailable. Error estimation is therefore an important integral part of numerical solution procedures.

Numerical solutions of fluid flow and heat transfer problems generally include three groups of errors (Lilek and Perić [88]):

- **Modelling errors** are defined as the difference between the actual flow and the exact solution of the mathematical model, describing the behaviour of the system in terms of coupled partial differential equations. In the case of laminar flows, modelling errors may be considered negligible for practical purposes – the Navier-Stokes equations represent a sufficiently accurate model of the flow. In the case of turbulent, two-phase or reacting flows, the additional models do not always describe the underlying physical processes accurately. In order to produce a “manageable” mathematical model certain simplifications are introduced in its construction, potentially causing high modelling errors. A better mathematical model requires a better understanding of the underlying physical processes, implies larger systems of equations and an increase in overall computational effort
- The second group of errors originates from the method used to solve the mathematical model. Considering the complexity of the problem and non-linearity of the equations, it is unreasonable to expect analytical solutions for all but simplest flow situations. We are forced to resort to an approximate numerical

solution method. **Discretisation errors** describe the difference between the exact solution of the system of algebraic equations obtained by discretising the governing equations on a given grid and the (usually unknown) exact solution of the mathematical model. Discretisation errors depend on the accuracy of the equation discretisation method, as well as the discretisation of the solution domain.

- The system of algebraic equations obtained from the discretisation is solved using an iterative solver. The difference between the approximate solution of the system obtained from the iterative solver and the exact solution of the system is described by the **iteration convergence errors**. They can be reduced to an arbitrary level, specified by the solver tolerance.

Most mathematical models require some kind of empirical input to calibrate the model constants. For this calibration, it is necessary to ensure that the discretisation and iteration convergence errors are sufficiently small. As the mathematical models become more and more accurate, the issue of discretisation accuracy becomes more important.

Having in mind the properties of the discretisation, it is possible to state several *a-priori* facts about the error. Numerical discretisation of a particular problem consists of two steps: discretisation of the solution domain and equation discretisation. In the first step, the solution domain is decomposed into discrete space and time intervals. In equation discretisation a variation of the variable over each region is prescribed, usually in a polynomial form. As the number of discrete regions increases to infinity, the approximate solution tends to the exact solution of the mathematical model. Alternatively, an increase in the order of interpolation leads to the same result. It is therefore possible to establish two ways of improving the accuracy of a numerical solution: increasing the number of computational points and increasing the order of interpolation.

The desired solution accuracy can be specified before the actual analysis it depends on the objective of the analysis and the accuracy of the mathematical

models used. If the solution is not accurate enough, the discretisation practice can be changed. Error estimation, on the other hand, requires a numerical solution in order to estimate the error. An adaptive procedure, producing the numerical solution of pre-determined accuracy will therefore consist of several numerical solutions, followed by error estimation and a suitable modification of the discretisation practice.

In this study, the Finite Volume method of discretisation has been coupled with an error-driven adaptive mesh refinement procedure in order to automatically produce numerical solutions of pre-determined accuracy. The procedure consists of a Finite Volume-type discretisation, followed by an *a-posteriori* error estimation tool and adaptive local mesh refinement algorithm. These parts interact automatically, without any user intervention. The adaptive procedure creates the solution that satisfies the accuracy criterion.

In the next Section an overview of the subject is presented, covering the relevant studies concerning the accuracy of Finite Volume discretisation, *a-posteriori* error estimation and adaptive refinement.

1.2 Previous and Related Studies

1.2.1 Convection Discretisation

The majority of fluid flows encountered in nature and industry are characterised by high Reynolds numbers, implying the dominance of convective effects (Hirsch [65]). While the fundamentals of the Finite Volume discretisation are well understood (Patankar [105], Hirsch [65]), discretisation of the convection term has been a subject of continual intense debate.

In the framework of the second-order accurate Finite Volume Method (FVM) a consistent discretisation scheme for the convection term would be second-order accurate Central Differencing (CD). However, the combination of the explicit time-integration, standard in the early development of numerical methods, and Cen-

tral Differencing creates an unconditionally unstable discretisation practice (Hirsch [65]). In order to achieve stability, first-order accurate differencing schemes have been introduced (Courant, Isaacson and Rees [33], Lax [78], Gentry *et al.* [50]). The unsatisfactory behaviour of first-order schemes has led Lax and Wendroff [79] to search for the second-order accurate discretisation. In the Lax-Wendroff family of schemes, stability is obtained by combining the spatial and temporal discretisation, leading to a variety of two-step (MacCormack [90], Lerat and Peyret [85]) and implicit schemes (MacCormack [91], Lerat [84]). In the case of steady-state calculations, the combined spatial and temporal discretisation introduces an unrealistic dependence of the solution on the time-step used to create it. In order to overcome this anomaly, a family of second-order schemes with independent time integration has been developed in the work of Beam and Warming [13, 14] and Jameson *et al.* [68]. Although this approach removes the dependence of spatial accuracy on the size of the time-step, the differencing schemes of the Beam and Warming family cause non-physical oscillations in the solution, severely reducing its quality. As a consequence, the numerical procedure can produce values of the dependent variable that are outside of its physically meaningful bounds. If one considers the transport of scalar properties common in fluid flow problems, such as phase fraction, turbulent kinetic energy, progress variable *etc.*, the importance of boundedness becomes clear. For example, a negative value of turbulent kinetic energy in calculations involving $k - \epsilon$ turbulence models results in negative viscosity, with disastrous effects on the solution algorithm. It is therefore essential to obtain bounded numerical solutions when solving transport equations for bounded properties.

The Beam and Warming family of schemes attempts to solve the boundedness problem by introducing a fourth-order artificial dissipation term (Hirsch [65]), but boundedness still cannot be guaranteed. Artificial diffusion terms, on the other hand, reduce the accuracy of the scheme, particularly in the regions of high gradients. The task of creating a good differencing scheme boils down to a balance between boundedness and accuracy.

An alternative view on the issues of accuracy and boundedness can be based on

the sufficient boundedness criterion for the system of algebraic equations. The only convection differencing scheme that guarantees boundedness is Upwind Differencing (UD), as all the coefficients in the system of algebraic equations will be positive even in the absence of physical diffusion (Patankar [105]). This is effectively done by introducing an excessive amount of numerical diffusion, which changes the nature of the problem from convection-dominated to convection-diffusion balanced. It was noted by several researchers (Boris and Book [20], Raithby [112, 114], Leonard [81]) that in cases of high streamline-to-grid skewness, this degradation of accuracy becomes unacceptable. Although, in principle, mesh refinement solves the problem, the necessary number of cells is totally impractical for engineering problems (Leonard [81]).

Several possible solutions to these problems have been proposed, falling into one of the following categories:

- Locally analytical schemes (LOADS by Raithby [148], Power-Law scheme by Patankar [105]) use the exact or approximate one-dimensional solution for the convection-diffusion equation in order to determine the face value of the dependent variable. Although bounded and somewhat less diffusive than UD, their accuracy in 2-D and 3-D is still inadequate.
- Upwind-biased differencing schemes, including first-order Upstream-weighted differencing by Raithby and Torrance [114], Linear Upwinding by Warming and Beam [146] and Leonard's QUICK differencing scheme [81]. The idea behind the upwind-biased schemes is to preserve the boundedness of UD by biasing the interpolation depending on the direction of the flux. The amount of numerical diffusion is somewhat smaller than for UD, but boundedness is not preserved.
- Skew-Upwind Differencing schemes Raithby [112, 113] owe their derivation to the fact that UD does not smear the solution in the case of mesh-to-flow alignment. It is therefore logical to create an upwind scheme that follows the direction of the flow, rather than the mesh. The resulting differencing scheme

behaves better than UD, but with better resolution also introduces unboundedness. Bounding of such schemes considerably reduces their accuracy, as in the case of Bounded Streamwise Upwinding (BSUD) of Gosman and Lai [55] and Sharif and Busnaina [122].

- **Switching schemes.** In his Hybrid Differencing scheme, Spalding [126], recognises that the sufficient boundedness criterion holds even for Central Differencing if the cell Peclet number is smaller than two. Under such conditions, Hybrid Differencing prescribes the use of CD, while UD is used for higher Pe -numbers in order to guarantee boundedness. However, in typical flow situations, the Pe -number is considerably higher than two and the scheme reduces to UD in the bulk of the domain.
- **Blended Differencing,** introduced by Perić [109]. Recognising the sufficient boundedness criterion as too strict for practical use, Perić proposes a “blending” approach, using a certain amount of upwinding combined with a higher-order scheme (CD or LUD) until boundedness is achieved. Although this approach potentially improves the accuracy, it is not known in advance how much blending should be used. In spite of the fact that the amount of blending needed to preserve boundedness varies from face to face, Perić proposes a constant blending factor for the whole mesh.

The quest for bounded and accurate differencing schemes continues with the concept of flux-limiting. Boris and Book [20] introduce a flux-limiter in their Flux Corrected Transport (FCT) differencing scheme, generalised for multi-dimensional problems by Zalesak [152]. The idea has been extensively used by van Leer in a series of papers working “Towards the ultimate conservative differencing scheme” [138, 139, 140, 141, 142]. These methods are sometimes classified as “shock-capturing schemes”, eventually resulting in the class of Total Variation Diminishing (TVD) differencing schemes. TVD schemes have been developed by Harten [58, 59], Roe [118], Chakravarthy and Osher [27] and others. A general procedure for constructing a TVD differencing scheme has been described by Osher and Chakravarthy [103].

Sweby [129] introduces a graphical interpretation of limiters (Sweby's diagram) and examines the accuracy of the method.

TVD methodology has been originally derived from the entropy condition for supersonic flows and subsequently extended to general scalar transport. The effective "blending factor" between the higher-order unbounded and first-order bounded differencing scheme depends on the local shape of the solution, thus introducing a non-linear dependence of the solution on itself. The convergence of this non-linear coupling to a unique solution can be strictly proven only for the explicit discretisation in one spatial dimension¹.

One of the main conclusions of the TVD analysis is that a differencing scheme has to be non-linear in order to be bounded and more than first-order accurate. TVD can be classified as a switching-blending methodology in which the discretisation practice depends on the local shape of the solution. It offers reasonably good accuracy and at the same time guarantees boundedness. It has been noted (Hirsch [65], Leonard [83]) that limiters giving good step-resolution, such as Roe's SUPERBEE [118] tend to distort smooth profiles. On the other hand, limiters such as MINMOD (Chakravarthy and Osher [27]), although being suitable for smooth profiles are still too diffusive.

In order to develop a differencing scheme that is able to give good resolution of sharp profiles and at the same time follow smooth profiles well, the Normalised Variable Approach (NVA) has been introduced by Leonard [82]. The TVD criterion has been rejected as too diffusive. The new condition requires local boundedness on a cell-by-cell basis. A series of differencing schemes based on the Normalised Variable Diagram (NVD) has been presented in recent years. The most popular are SHARP by Leonard, [82], SMART by Gaskell and Lau [49], UMIST by Lien and Leschziner [87] and Zhu's HLPD [153]. Leonard [83] introduces a general bounding method based on the NVD diagram. Unlike the TVD criterion, NVA does not offer any guarantee as regards the convergence of the differencing scheme, even on simple

¹The proof hinges on the fact that all explicit differencing schemes of the Lax-Wendroff and Beam-Warming type reduce to the upwind scheme. For details see *eg* Hirsch [65]

one-dimensional situations.

NVD differencing schemes produce remarkably good results for both stepwise profiles and smooth variations of the dependent variable. The amount of numerical diffusion is reduced to a minimum. However, as a result of the locally changing discretisation practice problems with convergence sometimes occur. A modified implementation proposed by Zhu [154] improves convergence, but boundedness can be guaranteed only for the converged solution.

Apart from the issues of accuracy and boundedness, which are essential for accurate calculations, modern differencing schemes are also required to be convergent and computationally inexpensive. The issue of computational cost includes both the additional face-by-face operations required to determine the weighting factors in TVD and NVD schemes and the additional effort required to obtain solutions for steady-state problems. With the development of NVD, the accuracy and boundedness of differencing schemes has been improved at the expense of convergence. For this reason, in author's opinion, the issue of convection discretisation is still not fully resolved.

1.2.2 Error Estimation

The use of error estimates as control parameters in numerical procedures is an old subject in numerical analysis. Automatic step-control and higher order predictor-corrector schemes in the numerical solution of ordinary differential equations have been standard tools for several decades.

The idea of using *a-posteriori* error estimates on the solutions of partial differential equations is more recent. In the Finite Element community the idea has been popularised by Babuska, Rheinboldt and their colleagues [8, 9, 10], Bank and Weiser [12], Oden *et al.* [99] and others. These efforts have been mainly directed at elliptic boundary value problems.

There is a wide range of popular error estimation procedures for Finite Element calculations. Oden *et al.* [98] present five groups of error estimators. These include Element- and Subdomain-Residual methods, Duality methods, Interpolation and

Post-processing methods. Element Residual methods use the residual in a numerical solution to estimate the local error. The residual is a function measuring how much the approximate solution fails to satisfy the governing differential equations and boundary conditions for the particular finite element. Duality methods, valid for self-adjoint elliptic problems, use the duality theory of convex optimisation to derive the upper and lower bounds of the error. Subdomain-Residual methods are based on the solution of the local error problem over a patch of finite elements. Interpolation methods use the interpolation theory of finite elements to produce a crude estimate of the leading term of the truncation error. Post-processing methods are based on the fact that the solution (which is expected to be smooth) can be improved by some smoothing algorithm. The estimate of the error is obtained by comparing the post-processed version of the solution with the one obtained from the actual calculation.

All these methods are strongly mathematically based and their properties have been examined for a wide range of shape functions. They have been used not only for symmetric boundary value problems but have also been extended to unisymmetric and convection-diffusion problems.

The Local Residual Problem Error estimate is the most recent error estimation method in the Finite Element method. It produces impressive results, consistently giving highly accurate estimates for a large variety of problems. It has been developed mainly by Ainsworth and Oden [2, 3, 4] and Ainsworth [1], but also includes the previous work by Bank and Weiser [11, 12] and Kelly [71]. The method has been extended to the Navier-Stokes problem in the work of Oden [101, 102]. It is based on the element residual method with elements of the duality theory. It is possible to show that this error estimate gives a strict upper bound on the solution error in the energy norm. It requires the solution of a local error problem over each finite element and an error flux equilibration procedure. Error flux equilibration has been discussed in length by Kelly [71] and Ainsworth and Oden [3]. Kelly shows that non-equilibrated fluxes result in gross over-estimation of the solution error. The analysis of the flux equilibration problem has been given by Ainsworth and Oden

[4]. Recent work of Oden *et al.* [102] presents an adaptive refinement technique based on this error estimate applied to incompressible Navier-Stokes equations.

Error estimation for the Finite Volume Method has been originally examined in conjunction with turbulence modelling (McGuirk *et al.* [93]). The main objective was to estimate the accuracy of the solution in absolute terms. In order to remove unphysical oscillations in the solution, the convection term of the Navier-Stokes equation has been discretised using Upwind Differencing. This introduces excessive amounts of numerical diffusion which interferes with the turbulent diffusion introduced by the turbulence model. Validation of turbulence models becomes a complicated task – it is not easy to determine how much of the additional diffusion comes from the model and how much should be attributed to inaccurate discretisation. McGuirk and Rodi [92] and McGuirk *et al.* [93] describe a technique for measuring the numerical diffusion of Upwind Differencing. The numerical diffusion term is then compared with other terms in the transport equation. The accuracy of the solution depends on the ratio of the numerical diffusion term and the largest physical term in the equation, called the Numerical Diffusion Ratio (NDR). It has been shown that some of the computational grids used for model evaluation were too coarse to be used to study the performance of turbulence models and that grid-independence studies were misleading. In a later work by Tattersall and McGuirk [130], the numerical diffusion estimate has been coupled with an adaptive node-movement technique. The method has been used to calculate separated flows around airfoils. It is interesting to note that the first mesh adaptation in the presented test case actually increased the solution error due to the loss of orthogonality and mesh-to-flow alignment.

Richardson extrapolation is by far the most popular error estimation method in Finite Volume calculations. It has been extensively used on a variety of situations, ranging from supersonic flows (Berger and Oliger [17], Berger and Collela [16], Berger and Jameson [19]) to incompressible problems (Thompson and Ferziger [134], Muzaferija [97]). In order to estimate the error, Richardson extrapolation uses two solutions of the same problem on two different grids. The method naturally couples with the use of multigrid acceleration techniques, where two solutions on grids

with different cell sizes are already available. Richardson extrapolation is the only method that can treat non-linearities of the problem, as it compares the solutions of the complete coupled systems (Muzaferija [97]). Provided that the meshes are fine enough, the accuracy of the error estimate is acceptable.

For industrial CFD problems, it is not always feasible to produce two solutions. In some cases, it might be necessary to use hundreds of thousands of cells just to represent the geometrical features of the computational domain, as in the case of internal combustion engine cooling systems, steam turbine stators *etc.*. Single-mesh single-run error estimates are therefore required.

Haworth *et al.* [61], Kern [72] and Muzaferija [97] present a new approach to the problem of error estimation. With the development of NVD differencing schemes, convection discretisation is becoming more and more accurate. The amount of numerical diffusion introduced in order to preserve the boundedness of the solution has been considerably decreased. As a consequence, errors from other sources, such as insufficient mesh resolution and mesh quality have become more important. In such cases, an error estimate based only on numerical diffusion cannot produce an accurate overall picture of the solution quality. It has become necessary to estimate the error in the case of full second-order accurate discretisation without any numerical diffusion at all. If the numerical diffusion error is still of interest, the error estimates can subsequently be modified to capture these effects as well.

Haworth *et al.* [61] propose the use of the cell to cell imbalances in angular momentum and kinetic energy to measure of the local solution error. The method has been tested on a transient flow problem in an axisymmetric internal combustion engine. Unfortunately, the complexity of the selected test case does not allow comparison of the error estimate with the “exact” error. Also, the method is not capable of estimating the absolute error levels. An extension of the same approach to higher moments of the variable has also been suggested but the results of this extension have not been reported to date.

Muzaferija [97] proposes a method of error estimation based on the higher derivatives of the solution. This method uses higher-order face interpolation to obtain

better estimates of the face values for the flow variables. The imbalance resulting from higher-order interpolation corresponds to the truncation error source of Phillips [110] and is consequently used to estimate the error for each cell. In order to determine the absolute error level, a suitable normalisation practice has been suggested. A second error estimator suggested in this work solves the transport equation for the solution error, with the aforementioned cell imbalance as the source term. The estimated error is compared with the “exact” error, obtained using a numerical solution on a very fine mesh. The method is slightly less accurate than Richardson extrapolation, but it provides a single-mesh measure of the error even in the absence of numerical diffusion and a means of estimating its magnitude.

The work of Kern [72] is mainly concerned with the formulation of an error estimator for transient Euler and Navier-Stokes equations. The analysis is performed for scalar hyperbolic equations in one and two spatial dimensions. In order to follow the development of the numerical error in time, an error evolution equation has been derived. Control volumes for the error evolution equation are staggered in space and time relative to the basic mesh. In order to stabilise the solution procedure for hyperbolic equations, a certain amount of numerical diffusion has been introduced either by the Godunov (upwind) differencing scheme, or through flux limiting. In a similar way to Muzaferija [97], more accurate face values for the flow variables are obtained using Central Differencing and used as the source in the error evolution equation. The method therefore measures the difference in the solution between the effective discretisation and the second-order accurate approximation, which is, in effect, numerical diffusion. The evolution equation for the error is extended to two-dimensional problems with constant and variable coefficients, as well as systems of differential equations. For equations with a diffusion term, the error source term is modified to include higher-order derivatives, taking into account the error in the diffusion term. Error estimation results are presented in terms of the Experimental Order of Convergence (EOC), representing the rate of reduction of the error with the number of cells. The accuracy of the method has not been tested against the analytical solution.

In comparison with the abundance of well-tested and reliable error estimators in the Finite Element field, Finite Volume error estimation is still in its early stages of development. The only well-examined and widely used method is Richardson extrapolation, which in turn requires two solutions of the same problem on two different meshes. A wide scope of ideas from the Finite Element field can, however, be modified for the use in the Finite Volume method, as will be demonstrated later in this Thesis.

1.2.3 Adaptive Refinement

In order to improve the accuracy of subsequent solutions, the distribution of the error can be used to introduce an appropriate change in the discretisation practice in the region of high error. In other parts of the domain, where the local error is considered to be sufficiently small, such change may not be necessary. The local changes in discretisation are commonly known as “mesh refinement”.

Mesh refinement strategies are usually divided into three groups, depending on the type of the change introduced in the discretisation.

- In **h-refinement** additional computational points are inserted locally in regions of high numerical error without disturbing the rest of the mesh. It is also possible to remove points from regions in which the error is low through an “unrefinement” procedure. Thus, the total number of points generally changes during the refinement/unrefinement process. The method is particularly suitable for problems with discontinuous solutions, requiring high local refinement. Examples of *h*-refinement can be found in the works of Coelho *et al.* [31], Vilsmeier and Hänel [145], Muzaferija [97] and others.
- **r-refinement** keeps the number of computational points constant throughout the calculation, but redistributes them depending on the distribution of the solution error. The structure of the mesh is preserved, which makes the method particularly interesting for single- or multi-block structured meshes. The main drawback of this approach is that it is not known in advance whether the

desired level of accuracy is obtainable with the available number of points. In cases where high local refinement is needed, r -refinement may cause high mesh distortion and severely degrade the mesh quality in regions where the resolution is not needed (Hawken *et al.* [60]). Point relocation algorithms are based on weighting functions derived from the error estimate. In order to perform the adaptation, the mesh is described as an elastic mass-spring system with weighting functions as the load. The distribution of points in the refined mesh corresponds to the locations of points under the load (see *eg.* Ramakrishnan [115]). Particular care has to be taken in order to prevent the mesh from overlapping. Examples of r -refinement can be found in Tattersall and McGuirk [130], Ramakrishnan [115], Dwyer [41] and Dandekar *et al.* [35].

- The third method of refinement is called **p-refinement**. It is particularly suitable for Finite Element calculations. This refinement procedure involves the use of higher-order shape functions in regions of high numerical error. As the higher-order finite element uses more computational nodes embedded in the original mesh, changes in the mesh structure and connectivity result. In order to close the system, additional coupling equations are required, complicating the form of the resulting system of algebraic equations and usually requiring much more computational effort for the solution (Rachowicz *et al.* [111]). The method is suitable for the problems with smoothly changing solutions. In the vicinity of steep gradients, the higher-order shape functions are prone to spurious oscillations even more than their lower-order counterparts. While p -refinement seems to be practical for Finite Element calculations, it has been rarely used outside of the Finite Element community. Calculations with p -refinement have been presented by *eg.* Zienkiewicz [156], Oden *et al.* [100] and others.

Reviews of adaptive techniques can be found in *eg.* Anderson [6], Thompson [133] and Hawken *et al.* [60].

In the framework of h -refinement for the Finite Volume method, several different

ways of point addition have been suggested, with different implications with respect to the solution accuracy and complexity of the flow solver. Early developments of adaptive grid techniques based on error estimation in the Finite Difference and Finite Volume methods were associated with the multigrid approach. The effort was directed towards the solution of the Euler and Navier-Stokes equations. Brandt [22] describes a coupled multigrid-local refinement method in which patches of refinement cover the regions where high resolution is needed. The method is referred to as the “segmental approach”. It has been further modified by Caruso [24, 25]. This type of method is used for transonic and supersonic flows with discontinuities. It uses a sequence of overlapping grids of increased fineness, thus allowing multiple levels of refinement. Each of the overlapping “patches” is an orthogonal structured grid which can be rotated relative to the basic grid. An optimisation procedure is used in order to determine the optimum number, size, relative distribution and orientation of the refinement “patches”. In the flow solver, each “patch” is treated independently, with the information transfer between the different parts of the mesh performed through the “patch” boundary conditions. The algorithm is computationally efficient since it deals with a series of uniform orthogonal structured grids, but the critical point is the transfer of information between the overlapping grids through boundary conditions. This is done explicitly, resulting in weaker coupling and slower convergence. Resolution problems have been reported at places where flow features intersect with “patch” boundaries (Berger [16, 17, 18]).

Segmental refinement procedure has been further modified by Berger and Oliger [17] and Berger and Collela [16], with an error estimation procedure based on Richardson extrapolation. The problem is solved on two grids with different cell sizes and with different time-step sizes for each patch. The difference in the solution is used to estimate the leading term of the truncation error. Since the meshes are structured, uniform and orthogonal, no additional storage is required. The coarser mesh is obtained by using every other point of the fine mesh. Cells in which the error is larger than some pre-determined value are then marked for refinement. A clustering algorithm developed by Berger [15] is used to optimise the construction

of patches, their position and mesh size. It uses concepts from pattern recognition and artificial intelligence theory. The simple optimisation algorithm is described in [17]. One of the attractive features of this approach is that it can be used for moving shocks in transient calculations although such a calculation has never been reported.

Thompson and Ferziger [134] presented an adaptive multigrid algorithm for steady-state Navier-Stokes equations coupled with the multigrid approach. The error estimation procedure is again based on Richardson extrapolation. Reductions of the CPU time and computer memory of 20 % and 40 % respectively, compared to the “pure” multigrid method have been reported. The adaptive refinement procedure is based on the work of Caruso [24, 25]. As a consequence of the interpolation procedure needed to determine boundary conditions on the refinement “patches”, the method does not guarantee local mass conservation until a converged solution is reached. A modified interpolation practice has been proposed but the problem has never been appropriately solved.

All these methods use structured orthogonal grids and are usually coupled with multigrid acceleration. The grids are superimposed on the basic grid and the calculation is coupled through the explicit update of “patch” boundary conditions.

Simpson [124] and Chen *et al.* [28] suggests the refinement procedure in which the refinement patches are embedded into the original mesh, thus removing the interpolation problem. The resulting mesh is then treated in a multi-block manner. Although this approach presents a considerable improvement in comparison with the earlier work, it is not appropriate for the situations with a large number of embedded refinement levels, as the number of blocks becomes so large that it significantly impairs the performance of the code (Chen *et al.* [28]).

A number of mesh refinement algorithms based on tetrahedral grids for Euler calculations have been proposed recently (*eg.* Vidwans and Kallinderis [70, 144], Sonar *et al.* [125]). Tetrahedral meshes offer geometrical flexibility, allow simple and highly localised refinement and can be created by automatic mesh generation procedures. Although this approach produces very good results in inviscid calculations, the ex-

tension of the method to viscous flows has been somewhat less successful. Vilsmeier and Hänel [145] have developed an adaptive Finite Volume algorithm on tetrahedral meshes for Euler and Navier-Stokes equations using h -refinement on a cell-by-cell basis. The emphasis has been placed on the improvement of mesh quality through successive refinement and anisotropic stretching. Virtual stretching of triangular elements has been introduced to provide the capability of mesh alignment. It is performed in the vicinity of walls and in regions of high shear, with the stretching direction determined from the gradients of the flow variables. Unfortunately, this results in high distortion of the mesh, decreasing the accuracy of the method.

Muzaferija [97] and Coelho *et al.* [31] present a method that combines the quality of hexahedral meshes with the capability of mesh refinement. Regions of local refinement are embedded into the original grid. The interaction between the coarse and fine mesh regions is done implicitly, using “split hexahedral” cells. A split hexahedron is a cell type, hexahedral in topology, which can have more than six faces. This approach guarantees local and global conservation after every iteration, but requires an appropriate addressing structure of the mesh.

All efforts reviewed above provide encouraging evidence of the usefulness of the adaptive refinement techniques, particularly for the flows with discontinuities. In general, adaptive refinement is at its best when the regions of high error cover only a small portion of the computational domain. The performance of adaptive algorithms for incompressible flows has been examined almost exclusively for laminar flow regimes. In spite of the encouraging results, extensions to turbulent flows (*eg.* Chen *et al.* [28]) reported to date are very rare.

1.3 Present Contributions

This study makes the following specific contribution to the field of Computational Fluid Dynamics:

- A Finite Volume discretisation technique applicable to arbitrarily unstructured meshes in three spatial dimensions is presented. The governing equations

are solved in real space, using a fixed Cartesian coordinate system, for both steady-state and transient problems. The discretisation practice is second-order accurate in space and time, making it suitable for second-order continuum mechanics problems. The computational mesh consists of non-uniform polyhedral control volumes, with a variable number of neighbours. This simplifies the problems of mesh generation for complex geometries, as well as adaptive mesh refinement. In order to preserve high efficiency of the method on vector and parallel computer architectures, a “face-addressing” implementation is used.

- In order to establish the sources of the discretisation error, a detailed analysis of the present discretisation technique is performed. Numerical diffusion terms resulting from the convection differencing scheme, discretisation of the transient term and mesh-induced errors are derived.
- In an attempt to increase the accuracy of the method, a new second-order bounded differencing scheme is proposed. The Gamma differencing scheme is formulated as a bounded version of Central Differencing, following the Normalised Variable Approach (Leonard [82]). The efficiency of the algorithm on non-orthogonal meshes is improved through the choice of decomposition of the face area vector into the orthogonal and non-orthogonal part, based on the stability of the resulting non-orthogonal treatment.
- The issue of *a-posteriori* error estimation is analysed in general terms. The requirements on an error estimate are outlined, as well as three approaches to error estimation. Following each of these approaches, three new error estimates are proposed:
 - The Direct Taylor Series Error estimate is based on the Taylor series truncation error analysis. Unlike the widely accepted Richardson extrapolation, this new error estimate can be used on a single-mesh solution. In spite of its asymptotic exactness, the accuracy of this error estimate

is not considered satisfactory.

- The Moment Error estimate derives the solution error from the imbalance in higher moments of the variable. In order to provide the error magnitude, the imbalance is normalised in an appropriate way.
- The Residual Error estimate measures the error from the inconsistency between the prescribed variation of the function over the control volume and the face interpolation practice, enabling the estimation of the error in the convection-diffusion part of the discretisation. A normalisation procedure is used to estimate the influence of the residual on the local solution accuracy.

The Residual error estimate is also extended to include the influence of the temporal error. The presented approach allows a separate evaluation of the spatial and temporal error, which can be used to select the optimal time-step size for a given spatial mesh.

- Ainsworth and Oden [1, 2, 3, 4] show a possible use of the residual in order to derive the strict upper bound on the error in the energy norm. Their approach is extended to the Finite Volume discretisation, with an appropriate error flux balancing procedure.

The accuracy of the proposed error estimates is tested on several problems with analytical solutions.

- A mesh adaptation technique is presented. Having in mind the quality of the available automatic mesh generators, the adaptive local mesh refinement algorithm has been preferred over adaptive mesh generation. The present technique enables both insertion and removal of the computational points, depending on the level of the local error.
- Finally, an automatic error-controlled adaptive local refinement unrefinement algorithm is assembled and successfully applied to several steady-state and transient laminar and turbulent flows

1.4 Thesis Outline

In Chapter 2, the governing equations of continuum mechanics are summarised, together with the constitutive relations for Newtonian fluids. A short overview of turbulence modelling techniques is also presented.

Chapter 3 presents a second-order accurate Finite Volume method on arbitrarily unstructured meshes. The setup of the computational mesh and the addressing structure are described. Discretisation of a general scalar transport equation is examined term by term. The Gamma differencing scheme is derived and tested on several simple situations. The numerical diffusion tensors for the discretisation errors are derived and their influence on the solution is illustrated on simple problems. The performance of the proposed non-orthogonality treatments is also presented. Finally, the segregated approach for the coupled systems of differential equations used in this study is described and a solution methodology for steady-state and transient turbulent flow calculations is summarised.

The issue of *a-posteriori* error estimation is addressed in Chapter 4. Several new methods of error estimation for both steady-state and transient problems are proposed. A Taylor series truncation error analysis is performed and its result is compared with Richardson extrapolation. An alternative form of Taylor series error estimation is proposed. The Moment Error estimate is derived from the transport equation for the second moment of the solution. Inconsistency between face interpolation and volume integration is used to define the residual as the measure of the local solution error, resulting in the Residual Error estimate. The concept is modified in order to include the temporal derivative, thus extending its applicability to transient calculations. The proposed formulation of the residual is used to extend the Local Problem Error estimate to Finite Volume discretisation. The performance of error estimates is examined on simple cases with analytical solutions.

Chapter 5 outlines algorithms for error-driven adaptive mesh generation and local mesh refinement. Their interaction with the estimated error distribution is examined. The selected algorithm, based on cell-by-cell embedded refinement and

unrefinement and “1-irregular” meshes is described. Comparison of uniform and adaptive refinement is performed on a simple test case.

In Chapter 6, the proposed solution procedure is applied on four test cases. Supersonic inviscid flow over a forward-facing step is used to test the performance of adaptive refinement/unrefinement in conjunction with an error indicator. The accuracy of error estimation is examined on steady-state two-dimensional flow over a hill, in both laminar and turbulent flow regimes. The interaction of the wall-functions and low- Re turbulence models with error estimation and mesh refinement is also examined. A three-dimensional turbulent flow over a swept backward-facing step has been selected to examine the overall performance of the error-controlled adaptive refinement approach in a 3-D situation. Finally, laminar vortex shedding behind a cylinder illustrates the capabilities of the algorithm for transient flow calculations.

Chapter 7 summarises the Thesis and offers some conclusions and suggestions for future research.

Chapter 2

Governing Equations

2.1 Governing Equations of Continuum Mechanics

The numerical procedure presented in this study deals with the problems of **continuum mechanics**. Characteristic length- and time-scales for such problems are considerably larger than the scales of the discrete structure of the matter. It is therefore possible to describe any macroscopic physical property of the matter as a continuous function in macroscopic coordinates (time, space).

We shall start by introducing the concept of the material derivative, describing the rate of change of the intensive physical property ϕ in time

$$\frac{d}{dt} \int_{V_M(t)} \rho \phi(\mathbf{x}, t) dV - \frac{\partial}{\partial t} \int_{V_M(t)} \rho \phi dV + \oint_{\partial V_M(t)} d\mathbf{S} \cdot (\rho \phi \mathbf{U}), \quad (2.1)$$

where $\mathbf{U}$ is the velocity vector and $d\mathbf{S}$ is the outward pointing unit normal on $\partial V_M(t)$. The rate of change of ϕ in V_M is equal to its volume and surface sources:

$$\frac{\partial}{\partial t} \int_{V_M(t)} \rho \phi dV + \oint_{\partial V_M(t)} d\mathbf{S} \cdot (\rho \phi \mathbf{U}) = \int_{V_M(t)} Q_V(\phi) dV + \oint_{\partial V_M(t)} d\mathbf{S} \cdot \mathbf{Q}_S(\phi), \quad (2.2)$$

or, in the differential form:

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho \phi \mathbf{U}) = Q_V(\phi) + \nabla \cdot \mathbf{Q}_S(\phi). \quad (2.3)$$

The governing equations of continuum mechanics can be written in the form of Eqn. (2.3) (Aris [7]):

- Conservation of mass

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0, \quad (2.4)$$

- Conservation of linear momentum

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = \rho \mathbf{g} + \nabla \cdot \sigma, \quad (2.5)$$

- Conservation of angular momentum

$$\frac{\partial \rho (\mathbf{x} \times \mathbf{U})}{\partial t} + \nabla \cdot [\rho (\mathbf{x} \times \mathbf{U}) \mathbf{U}] = \rho (\mathbf{x} \times \mathbf{g}) + \mathbf{x} \times (\nabla \cdot \sigma). \quad (2.6)$$

Combination of Eqs. (2.5) and (2.6) states that σ is a symmetric tensor:

$$\sigma = \sigma^T, \quad (2.7)$$

- Conservation of energy

$$\frac{\partial \rho e}{\partial t} + \nabla \cdot (\rho e \mathbf{U}) = \rho \mathbf{g} \cdot \mathbf{U} + \nabla \cdot (\sigma \cdot \mathbf{U}) - \nabla \cdot \mathbf{q} + \rho Q, \quad (2.8)$$

- The entropy inequality

$$\frac{\partial \rho s}{\partial t} + \nabla \cdot (\rho s \mathbf{U}) \geq \nabla \cdot \left(\frac{\mathbf{q}}{T} \right) + \frac{\rho Q}{T}, \quad (2.9)$$

where

- ρ is the density,
- $\mathbf{x}$ is the position vector,
- σ is the stress tensor,
- e is the total specific energy,
- Q is the volume energy source
- s is the specific entropy

- T is the temperature and
- $\mathbf{q}$ is the heat flux.

The above conservation laws are valid for any continuum. The number of unknown quantities is, however, larger than the number of equations in the system, making the system indeterminate.

2.2 Constitutive Relations for Newtonian Fluids

In order to close the system, it is necessary to introduce additional, so-called **constitutive relations**. They depend on the properties of the continuous medium in question. In the case of Newtonian fluids, the following set of constitutive relations is used:

- The internal energy equation, defining the internal energy as a function of pressure P and temperature T :

$$u = u(P, T). \quad (2.10)$$

The total energy is calculated as the sum of the kinetic e_M and internal energy (and all others, like chemical, nuclear *etc.* which will be neglected in this study):

$$e = e_M + u(P, T) = \frac{1}{2} \mathbf{U} \cdot \mathbf{U} + u(P, T), \quad (2.11)$$

- The equation of state:

$$\rho = \rho(P, T), \quad (2.12)$$

- The Fourier's law of heat conduction:

$$\mathbf{q} = -\lambda \nabla T, \quad (2.13)$$

- Generalised form of the Newton's law of viscosity:

$$\sigma = - \left(P + \frac{2}{3} \mu \nabla \cdot \mathbf{U} \right) \mathbf{I} + \mu \left[\nabla \mathbf{U} + (\nabla \mathbf{U})^T \right]. \quad (2.14)$$

These constitutive relations, together with the governing equations for a continuum create a closed system of partial differential equations for Newtonian fluids:

- Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0, \quad (2.15)$$

- Navier-Stokes equations:

$$\begin{aligned} \frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = & \rho \mathbf{g} - \nabla \left(P + \frac{2}{3} \mu \nabla \cdot \mathbf{U} \right) \\ & + \nabla \cdot \left[\mu \left(\nabla \mathbf{U} + (\nabla \mathbf{U})^T \right) \right], \end{aligned} \quad (2.16)$$

- Energy equation:

$$\begin{aligned} \frac{\partial \rho e}{\partial t} + \nabla \cdot (\rho e \mathbf{U}) = & \rho \mathbf{g} \cdot \mathbf{U} - \nabla \cdot (P \mathbf{U}) - \nabla \cdot \left(\frac{2}{3} \mu (\nabla \cdot \mathbf{U}) \mathbf{U} \right) \\ & + \nabla \cdot \left[\mu \left(\nabla \mathbf{U} + (\nabla \mathbf{U})^T \right) \cdot \mathbf{U} \right] + \nabla \cdot (\lambda \nabla T) + \rho Q, \end{aligned} \quad (2.17)$$

- The entropy inequality:

$$\frac{\partial \rho s}{\partial t} + \nabla \cdot (\rho s \mathbf{U}) \geq \nabla \cdot \left(\frac{-\lambda \nabla T}{T} \right) + \frac{\rho Q}{T}. \quad (2.18)$$

The transport coefficients λ and μ are also functions of the thermodynamic state variables:

$$\lambda = \lambda(P, T), \quad (2.19)$$

$$\mu = \mu(P, T). \quad (2.20)$$

Transport equations for the internal and kinetic energy have the following form (Aris [7]):

$$\begin{aligned} \frac{\partial \rho u}{\partial t} + \nabla \cdot (\rho u \mathbf{U}) = & \nabla \mathbf{U} : \left[\mu \left(\nabla \mathbf{U} + (\nabla \mathbf{U})^T \right) - \left(\frac{2}{3} \mu \nabla \cdot \mathbf{U} \right) \mathbf{I} \right] \\ & - P \nabla \cdot \mathbf{U} + \nabla \cdot (\lambda \nabla T) + \rho Q, \end{aligned} \quad (2.21)$$

$$\begin{aligned} \frac{\partial \rho e_M}{\partial t} + \nabla \cdot (\rho e_M \mathbf{U}) = & \rho \mathbf{g} \cdot \mathbf{U} - \nabla \cdot (P \mathbf{U}) + P \nabla \cdot \mathbf{U} \\ & - \nabla \cdot \left(\frac{2}{3} \mu (\nabla \cdot \mathbf{U}) \mathbf{U} \right) + \nabla \cdot \left[\mu \left(\nabla \mathbf{U} + (\nabla \mathbf{U})^T \right) \cdot \mathbf{U} \right] \\ & - \nabla \mathbf{U} : \left[\mu \left(\nabla \mathbf{U} + (\nabla \mathbf{U})^T \right) - \left(\frac{2}{3} \mu \nabla \cdot \mathbf{U} \right) \mathbf{I} \right]. \end{aligned} \quad (2.22)$$

For incompressible isothermal fluids ($\rho = \text{const.}$, $\lambda = \infty$), the system can be further simplified:

$$\nabla \cdot \mathbf{U} = 0, \quad (2.23)$$

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{U}\mathbf{U}) = \mathbf{g} - \nabla p + \nabla \cdot (\nu \nabla \mathbf{U}), \quad (2.24)$$

where ν is the kinematic viscosity and p kinematic pressure. All other equations in the system are decoupled.

Most dependent variables require a gradient-diffusion term of the following form (Patankar [105]):

$$-\nabla \cdot (\Gamma_\phi \nabla \phi). \quad (2.25)$$

This term will therefore be included into the transport equation for a general tensorial property χ :

$$\frac{\partial \rho \chi}{\partial t} + \nabla \cdot (\rho \chi \mathbf{U}) - \nabla \cdot (\rho \Gamma_\chi \nabla \chi) = S_\chi(\chi), \quad (2.26)$$

where $S_\chi(\chi)$ contains all terms that cannot be fitted on the left-hand side.

2.3 Turbulence Modelling

Most fluid flows occurring in nature are turbulent. Turbulence can be described as a state of continuous instability in the flow, where it is still possible to separate the fluctuations from the mean flow properties. It is characterised by irregularity in the flow, increased diffusivity and energy dissipation (Tennekes and Lumley [132]). Turbulent flows are always three-dimensional and time dependent, even if the boundary conditions of the flow do not change in time. The range of scales in such flows is very large, from the smallest turbulent eddies characterised by Kolmogorov microscales, to the flow features comparable with the size of the geometry. A comprehensive review of simulation techniques for turbulent flows can be found in Ferziger [47]. A brief overview of the modelling techniques will be given here.

There are several possible approaches to the simulation of turbulent flows. The first, Direct Numerical Simulation (DNS) (Eswaran and Pope [44, 45], Rogallo and

Moin [119], Rogers and Moin [120]) numerically integrates the governing equations over the whole range of turbulent scales. The requirements on mesh resolution and time-step size put very high demands on the computer resources, rendering it unsuitable for engineering applications.

The second approach is generally known as Large Eddy Simulation (LES). In order to separate different length scales in a turbulent flow field, a spatial filter is applied. Large scale structures that can be resolved by the numerical method on a given mesh are called the super-grid scales. The influence of all other (sub-grid) scales to the super-grid behaviour is modelled. The rationale behind this principle lies in the fact that the small scales of turbulence are more homogeneous and isotropic and therefore easier to model. As the mesh gets finer, the number of scales that require modelling becomes smaller, thus approaching the Direct Numerical Simulation. Examples of this approach can be found in Deardorff [38], Givi [53] and Moin and Kim [95].

An alternative approach to the simulation of turbulent flows is statistical. Separating the local value of the variable into the mean and the fluctuation around the mean, it is possible to derive the equations for the mean properties themselves. The selection of averaging methods depends on the characteristics of turbulent flow, and according to Hinze [64], the following three averaging method can be distinguished:

1. Time averaging in a fixed point of space, for stationary turbulence,
2. Space averaging for a fixed moment in time in the case of homogeneous turbulence,
3. Ensemble averaging for a series of identical experiments. This is the most general form of averaging.

All these methods can appear in two versions, unweighted (Reynolds) and weighted (*eg* density-weighted Favre averaging, [46]). The Reynolds averaging technique operates as follows:

$$\phi(\mathbf{x}, t) = \bar{\phi}(\mathbf{x}, t) + \phi'(\mathbf{x}, t), \quad (2.27)$$

where $\phi'(\mathbf{x}, t)$ denotes the fluctuation about the mean value, defined as:

$$\bar{\phi}(\mathbf{x}, t) = \lim_{\mathcal{N} \rightarrow \infty} \frac{1}{\mathcal{N}} \sum_{i=1}^{\mathcal{N}} \phi_i(\mathbf{x}, t), \quad (2.28)$$

and $\mathcal{N}$ is the number of identically performed experiments.

Applying the above averaging procedure to the incompressible Navier-Stokes equations, the following form of the averaged equations can be obtained:

$$\nabla \cdot \bar{\mathbf{U}} = 0, \quad (2.29)$$

$$\frac{\partial \bar{\mathbf{U}}}{\partial t} + \nabla \cdot (\bar{\mathbf{U}} \bar{\mathbf{U}}) = \mathbf{g} - \nabla \bar{p} + \nabla \cdot (\nu \nabla \bar{\mathbf{U}}) + \overline{\mathbf{U}' \mathbf{U}'}. \quad (2.30)$$

In the case of compressible flows, Favre averaging is usually applied (see *eg.* Favre [46], Cebeci and Smith [26]).

The term $\overline{\mathbf{U}' \mathbf{U}'}$ is called the Reynolds stress tensor. In order to close the system, further modelling is necessary.

The task of Reynolds averaged turbulence modelling is to express the Reynolds stress tensor in terms of the known quantities. There are two widely accepted approaches. The first approach formulates and solves the transport equation for the Reynolds stress tensor (and higher moments, depending on the order of the closure). It is, however, still necessary to model some of the terms, since the number of unknowns increases faster than the number of equations. Examples of this treatment can be found in Rotta [121], Launder *et al.* [75] and Hanjalić and Rodi [56]. The second, and more popular approach prescribes a relationship between the Reynolds stress and mean velocity gradient. Although models prescribing non-linear relations have been proposed recently (Speziale [127] and Shih *et al.* [123]), the most popular approach is to use the Boussinesq approximation (Boussinesq [21]), which prescribes a linear relation of the form:

$$\overline{\mathbf{U}' \mathbf{U}'} = \nu_t (\nabla \mathbf{U} + (\nabla \mathbf{U})^T) + \frac{2}{3} k \mathbf{I}, \quad (2.31)$$

where

$$k = \frac{1}{2} \overline{\mathbf{U}' \cdot \mathbf{U}'}. \quad (2.32)$$

The kinematic eddy viscosity ν_t can be evaluated in many different ways, ranging from algebraic relations and local equilibrium assumptions to the solution of transport equations. The most popular approach is to express ν_t as a function of the turbulent kinetic energy k and its dissipation rate ϵ , leading to a “two-equation” turbulence model:

$$\nu_t = C_\mu \frac{k^2}{\epsilon}, \quad (2.33)$$

$$\epsilon = \overline{\mathbf{U}'\mathbf{U}'} : \nabla \mathbf{U}', \quad (2.34)$$

so named because k and ϵ are obtained by the solution of their respective transport equations. The first model of this kind has been proposed by Harlow and Nakayama [57]. Derivation of the transport equations for k and ϵ can be found in *eg.* Launder and Spalding [77].

In fact, a wide variety of $k - \epsilon$ models exists, the most noteworthy being the standard $k - \epsilon$ model by Launder and Spalding [77] and the RNG $k - \epsilon$ variant by Yakhot and Orszag [150, 151].

The physics of turbulence in the vicinity of impermeable no-slip walls is considerably different from the other parts of the flow. It is therefore necessary to use appropriate turbulence models in the near-wall region. For the most general and detailed treatment, low- Re versions should be used (see *eg.* Lam and Bremhorst [74], Launder and Sharma [76] *etc.*). However, in order to resolve the near-wall details well, the computational mesh needs to be very fine in this region.

It is possible to compensate for the existence of the wall without resolving the near-wall region, albeit at the expense of considerable approximation and, as will be shown later, also with adverse effects on numerical resolution). Wall-functions Launder and Spalding [77] represent a simplified model of turbulence, which mimics the near-wall behaviour of the velocity, k and ϵ . It assumes that the flow near the solid wall behaves like a fully developed turbulent boundary layer. In numerical simulations, this model is used to bridge the regions of high gradients near the wall and couples with the high- Re $k - \epsilon$ model in the rest of the domain. For the details

of its derivation and implementation, the reader is referred to Launder and Spalding [77] and Gosman and Ideriah [54].

Chapter 3

Finite Volume Discretisation

3.1 Introduction

The purpose of any discretisation practice is to transform one or more partial differential equations into a corresponding system of algebraic equations. The solution of this system produces a set of values which correspond to the solution of the original equations at some pre-determined locations in space and time, provided certain conditions, to be defined later, are satisfied. The discretisation process can be divided into two steps. the discretisation of the solution domain and equation discretisation (Hirsch [65], Muzaferija [97]).

The discretisation of the solution domain produces a numerical description of the computational domain, including the positions of points in which the solution is sought and the description of the boundary. The space is divided into a finite number of discrete regions, called control volumes or cells. For transient simulations, the time interval is also split into a finite number of time-steps. Equation discretisation gives an appropriate transformation of terms of governing equations into algebraic expressions.

This Chapter presents the Finite Volume method (FVM) of discretisation, with the following properties:

- The method is based on discretising the integral form of governing equations

over each control volume. The basic quantities, such as mass and momentum, will therefore be conserved at the discrete level.

- Equations are solved in a fixed Cartesian coordinate system on the mesh that does not change in time. The method is applicable to both steady-state and transient calculations.
- The control volumes can be of a general polyhedral shape, with a variable number of neighbours, thus creating an arbitrarily unstructured mesh. All dependent variables share the same control volumes, which is usually called the collocated or non-staggered variable arrangement (Rhie and Chow [117], Perić [109]).
- Systems of partial differential equations are treated in the segregated way (Patankar and Spalding [107], van Doormaal and Raithby [137]), meaning that they are solved one at a time, with the inter-equation coupling treated in the explicit manner. Non-linear differential equations are linearised before the discretisation and the non-linear terms are lagged.

The details of the discretisation practice and implementation of boundary conditions will be described in the remainder of this Chapter. In Section 3.2 the mesh arrangement is described. Section 3.3 presents a discretisation procedure for a scalar transport equation. The discretisation of the convection, diffusion and source terms is presented in Section 3.3.1. Several possibilities for the discretisation of the temporal derivative are given in Section 3.3.2. The implementation of the boundary conditions is discussed in Section 3.3.3. A new second order accurate and bounded convection differencing scheme is proposed in Section 3.4. Some details of the solution procedure, including the treatment of mesh non-orthogonality, deferred correction and the under-relaxation are given in Section 3.5. Section 3.6 discusses the errors introduced during the discretisation process. Numerical examples, illustrating the behaviour of the discretised terms and numerical errors, are presented in Section 3.7.

The solution procedure for systems of partial differential equations requires special attention. The generalised segregated approach for pressure-velocity coupling is described in Section 3.8. Finally, some closing remarks are given in Section 3.9.

3.2 Discretisation of the Solution Domain

Discretisation of the solution domain produces a computational mesh on which the governing equations are subsequently solved. It also determines the positions of points in space and time where the solution is sought. The procedure can be split into two parts: discretisation of time and space.

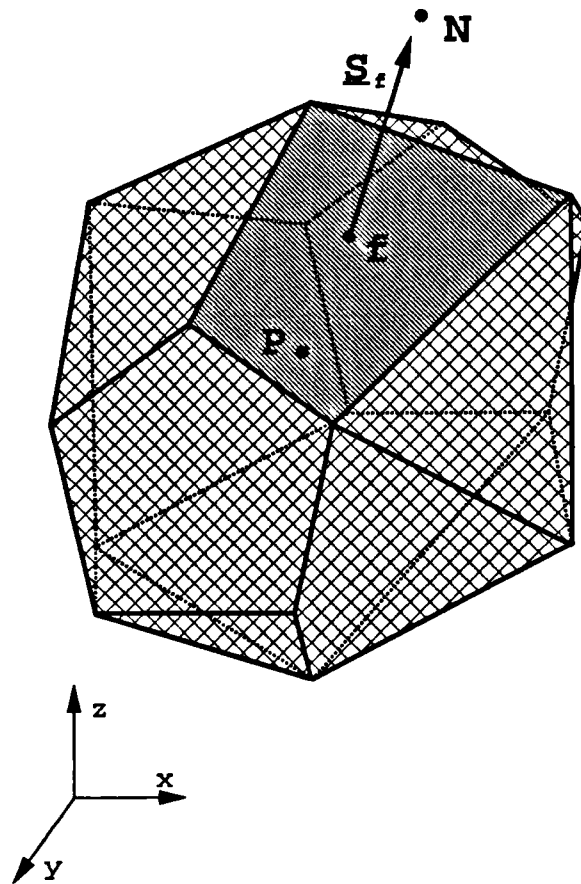


Figure 3.1: Control volume.

Since time is a parabolic coordinate (Patankar [105]), the solution is obtained by marching in time from the prescribed initial condition. For the discretisation of

time, it is therefore sufficient to prescribe the size of the time-step that will be used during the calculation.

The discretisation of space for the Finite Volume method used in this study requires a subdivision of the domain into control volumes (CV). Control volumes do not overlap and completely fill the computational domain. In the present study all variables share the same CV-s.

A typical control volume is shown in Fig. 3.1. The computational point P is located at the centroid of the control volumes, such that:

$$\int_{V_P} (\mathbf{x} - \mathbf{x}_P) dV = \mathbf{0}. \quad (3.1)$$

The control volume is bounded by a set of flat faces and each face is shared with only one neighbouring CV. The topology of the control volume is not important – it is a general polyhedron.

The cell faces in the mesh can be divided into two groups – internal faces (between two control volumes) and boundary faces, which coincide with the boundaries of the domain. The face area vector $\mathbf{S}_f$ is constructed for each face in such a way that it points outwards from the cell with the lower label, is normal to the face and has the magnitude equal to the area of the face. The cell with the lower label is called the “owner” of the face – its label is stored in the “owner” array. The label of the other cell “neighbour” is stored in the “neighbour” array. Boundary face area vectors point outwards from the computational domain – boundary faces are “owned” by the adjacent cells. For the shaded face in Fig. 3.1, the owner and neighbour cell centres are marked with P and N , as the face area vector $\mathbf{S}_f$ points outwards from the P control volume. For simplicity, all faces of the control volume will be marked with f , which also represents the point in the middle of the face (see Fig. 3.1).

The capability of the discretisation practice to deal with arbitrary control volumes gives considerable freedom in mesh generation. It is particularly useful in meshing complex geometrical configurations in three spatial dimensions. Arbitrarily unstructured meshes also interact well with the concept of local grid refinement, where the computational points are added in the parts of the domain where high

resolution is necessary, without disturbing the rest of the mesh.

3.3 Discretisation of the Transport Equation

The standard form of the transport equation for a scalar property ϕ is:

$$\underbrace{\frac{\partial \rho \phi}{\partial t}}_{\text{temporal derivative}} + \underbrace{\nabla \cdot (\rho \mathbf{U} \phi)}_{\text{convection term}} - \underbrace{\nabla \cdot (\rho \Gamma_\phi \nabla \phi)}_{\text{diffusion term}} = \underbrace{S_\phi(\phi)}_{\text{source term}} \quad (3.2)$$

This is a second-order equation, as the diffusion term includes the second derivative of ϕ in space. For good accuracy, it is necessary for the order of the discretisation to be equal to or higher than the order of the equation that is being discretised. The discretisation practice adopted in this study is second-order accurate in space and time and will be presented in the rest of this Chapter. The individual terms of the transport equation will be treated separately.

In certain parts of the discretisation it is necessary to relax the accuracy requirement, either to accommodate for the irregularities in the mesh structure or to preserve the boundedness of the solution¹. Any deviation from the prescribed order of accuracy creates a discretisation error, which is of the order of other terms in the original equation and disappears only in the limit of excessively fine mesh. Particular attention will therefore be paid to the sources of discretisation error representing such behaviour.

The accuracy of the discretisation method depends on the assumed variation of the function $\phi = \phi(\mathbf{x}, t)$ in space and time around the point P . In order to obtain a second-order accurate method, this variation must be linear in both space and time, *i.e.* it is assumed that:

$$\phi(\mathbf{x}) = \phi_P + (\mathbf{x} - \mathbf{x}_P) \cdot (\nabla \phi)_P, \quad (3.3)$$

$$\phi(t + \Delta t) = \phi^t + \Delta t \left(\frac{\partial \phi}{\partial t} \right)^t, \quad (3.4)$$

¹These issues will be addressed in length in Sections 3.3.1 and 3.6.

where

$$\phi_P = \phi(\mathbf{x}_P), \quad (3.5)$$

$$\phi^t = \phi(t). \quad (3.6)$$

Let us consider the Taylor series expansion in space of a function around the point $\mathbf{x}$:

$$\begin{aligned} \phi(\mathbf{x}) &= \phi_P + (\mathbf{x} - \mathbf{x}_P) \cdot (\nabla \phi)_P + \frac{1}{2} (\mathbf{x} - \mathbf{x}_P)^2 : (\nabla \nabla \phi)_P \\ &\quad + \frac{1}{3!} (\mathbf{x} - \mathbf{x}_P)^3 :: (\nabla \nabla \nabla \phi)_P \\ &\quad + \dots + \frac{1}{n!} (\mathbf{x} - \mathbf{x}_P)^n \underbrace{:::}_{n} \underbrace{(\nabla \nabla \dots \nabla \phi)_P}_{n} + \dots \end{aligned} \quad (3.7)$$

The expression $(\mathbf{x} - \mathbf{x}_P)^n$ in Eqn. (3.7) and consequent equations in this study represents the n th tensorial product of the vector $(\mathbf{x} - \mathbf{x}_P)$ with itself, producing an n th rank tensor. The operator ‘ $\underbrace{:::}_n$ ’ is the inner product of two n th rank tensors, creating a scalar.

Comparison between the assumed variation, Eqn. (3.3), and the Taylor series expansion, Eqn. (3.7), shows that the first term of the truncation error scales with $(\mathbf{x} - \mathbf{x}_P)^2$, which is for a 1-D situation equal to the square of the size of the control volume. The assumed spatial variation is therefore second-order accurate in space. An equivalent analysis shows that the truncation error in Eqn. (3.4) scales with Δt^2 , resulting in the second-order temporal accuracy.

The Finite Volume method requires that Eqn. (3.2) is satisfied over the control volume V_P around the point P in the integral form:

$$\begin{aligned} \int_t^{t+\Delta t} \left[\frac{\partial}{\partial t} \int_{V_P} \rho \phi dV + \int_{\Gamma_P} \nabla \cdot (\rho \mathbf{U} \phi) dV - \int_{V_P} \nabla \cdot (\rho \Gamma_\phi \nabla \phi) dV \right] dt \\ = \int_t^{t+\Delta t} \left(\int_{V_P} S_\phi(\phi) dV \right) dt. \end{aligned} \quad (3.8)$$

The discretisation of Eqn. (3.8) will now be examined term by term.

3.3.1 Discretisation of Spatial Terms

Let us first examine the discretisation of spatial terms. The generalised form of Gauss’ theorem will be used throughout the discretisation procedure, involving the e

identities:

$$\int_V \nabla \cdot \mathbf{a} dV = \oint_{\partial V} d\mathbf{S} \cdot \mathbf{a}, \quad (3.9)$$

$$\int_V \nabla \phi dV = \oint_{\partial V} d\mathbf{S} \phi, \quad (3.10)$$

$$\int_V \nabla \mathbf{a} dV = \oint_{\partial V} d\mathbf{S} \mathbf{a}, \quad (3.11)$$

where ∂V is the closed surface bounding the volume V and $d\mathbf{S}$ represents an infinitesimal surface element with associated outward pointing normal on ∂V .

A series of volume and surface integrals needs to be evaluated. Taking into account the prescribed variation of ϕ over the control volume P , Eqn. (3.3), it follows:

$$\begin{aligned} \int_{V_P} \phi(\mathbf{x}) dV &= \int_{V_P} [\phi_P + (\mathbf{x} - \mathbf{x}_P) \cdot (\nabla \phi)_P] dV \\ &= \phi_P \int_{V_P} dV + \left[\int_{V_P} (\mathbf{x} - \mathbf{x}_P) dV \right] \cdot (\nabla \phi)_P \\ &= \phi_P V_P, \end{aligned} \quad (3.12)$$

where V_P is the volume of the cell. The second integral in Eqn. (3.12) is equal to zero because the point P is the centroid of the control volume.

Let us now consider the terms under the divergence operator. Having in mind that the CV is bounded by a series of flat faces, Eqn. (3.9) can be transformed into a sum of integrals over all faces:

$$\begin{aligned} \int_{V_P} \nabla \cdot \mathbf{a} dV &= \oint_{\partial V_P} d\mathbf{S} \cdot \mathbf{a} \\ &= \sum_f \left(\int_f d\mathbf{S} \cdot \mathbf{a} \right). \end{aligned} \quad (3.13)$$

The assumption of linear variation of ϕ leads to the following expression for the face integral in Eqn. (3.13):

$$\begin{aligned} \int_f d\mathbf{S} \cdot \mathbf{a} &= \left(\int_f d\mathbf{S} \right) \cdot \mathbf{a}_f + \left[\int_f d\mathbf{S} (\mathbf{x} - \mathbf{x}_f) \right] : (\nabla \mathbf{a})_f \\ &= \mathbf{S} \cdot \mathbf{a}_f. \end{aligned} \quad (3.14)$$

Combining Eqs. (3.12), (3.13) and (3.14), a second-order accurate discretised form

of the Gauss' theorem is obtained:

$$(\nabla \cdot \mathbf{a}) V_P = \sum_f \mathbf{S} \cdot \mathbf{a}_f. \quad (3.15)$$

Here, the subscript f implies the value of the variable (in this case, $\mathbf{a}$) in the middle of the face and $\mathbf{S}$ is the outward-pointing face area vector. In the current mesh structure, the face area vector $\mathbf{S}_f$ point outwards from P only if f is “owned” by P . For the “neighbouring” faces $\mathbf{S}_f$ points inwards, which needs to be taken into account in the sum in Eqn. (3.15). The sum over the faces is therefore split into sums over “owned” and “neighbouring” faces:

$$\sum_f \mathbf{S} \cdot \mathbf{a}_f = \sum_{\text{owner}} \mathbf{S}_f \cdot \mathbf{a}_f - \sum_{\text{neighbour}} \mathbf{S}_f \cdot \mathbf{a}_f. \quad (3.16)$$

This is true for every summation over the faces. In the rest of the text, this split is automatically assumed.

3.3.1.1 Convection Term

The discretisation of the convection term is obtained using Eqn. (3.15):

$$\begin{aligned} \int_{V_P} \nabla \cdot (\rho \mathbf{U} \phi) dV &= \sum_f \mathbf{S} \cdot (\rho \mathbf{U} \phi)_f \\ &= \sum_f \mathbf{S} \cdot (\rho \mathbf{U})_f \phi_f \\ &= \sum_f F \phi_f, \end{aligned} \quad (3.17)$$

where F in Eqn. (3.17) represents the mass flux through the face:

$$F = \mathbf{S} \cdot (\rho \mathbf{U})_f. \quad (3.18)$$

The calculation of these face fluxes will later be discussed separately in Section 3.8. For now it can be assumed that the flux is calculated from the interpolated values of ρ and $\mathbf{U}$.

Eqs. 3.17 and (3.18) also require the face value of the variable ϕ calculated from the values in the cell centres, which is obtained using the convection differencing scheme.

Before we continue with the formulation of the convection differencing scheme, it is necessary to examine the physical properties of the convection term. Irrespective of the distribution of the velocity in the domain, the convection term does not violate the bounds of ϕ given by its initial distribution. If, for example, ϕ initially varies between 0 and 1, the convection term will never produce the values of ϕ that are lower than zero or higher than unity. Considering the importance of boundedness in the transport of scalar properties of interest (see Section 1.2.1), it is essential to preserve this property in the discretised form of the term.

3.3.1.2 Convection Differencing Scheme

The role of the convection differencing scheme is to determine the value of ϕ on the face from the values in the cell centres. In the framework of arbitrarily unstructured meshes, it would be impractical to use any values other than ϕ_P and ϕ_N , because of the storage overhead associated with the additional addressing information. We shall therefore limit ourselves to differencing schemes using only the nearest neighbours of the control volume.

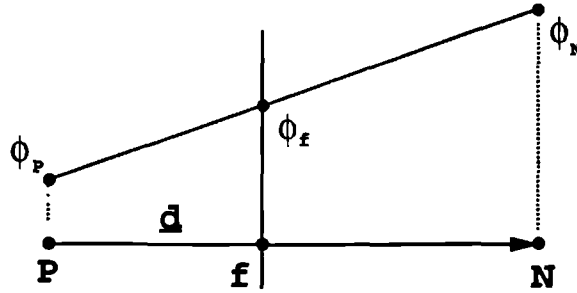


Figure 3.2: Face interpolation.

Assuming the linear variation of ϕ between P and N , Fig. 3.2, the face value is calculated according to:

$$\phi_f = f_x \phi_P + (1 - f_x) \phi_N. \quad (3.19)$$

Here, the interpolation factor f_x is defined as the ratio of distances $\overline{fN}$ and $\overline{PN}$:

$$f_x = \frac{\overline{fN}}{\overline{PN}}. \quad (3.20)$$

The differencing scheme using Eqn. (3.19) to determine the face value of ϕ is called **Central Differencing** (CD). Although this has been the subject of some debate, Ferziger and Perić [48] show that it is second order accurate even on non-uniform meshes. This is consistent with the overall accuracy of the method. It has been noted, however, that CD causes unphysical oscillations in the solution for convection-dominated problems (Patankar, [105], Hirsch [65]), thus violating the boundedness of the solution.

An alternative discretisation scheme that guarantees boundedness is **Upwind Differencing** (UD). The face value of ϕ is determined according to the direction of the flow:

$$\phi_f = \begin{cases} \phi_f = \phi_P & \text{for } F \geq 0. \\ \phi_f = \phi_N & \text{for } F < 0. \end{cases} \quad (3.21)$$

Boundedness of the solution is guaranteed through the sufficient boundedness criterion for systems of algebraic equations (see *eg.* Patankar, [105]). As it will be shown later (Section 3.6), boundedness of UD is effectively insured at the expense of the accuracy, by implicitly introducing the numerical diffusion term. This term violates the order of accuracy of the discretisation and can severely distort the solution.

Blended Differencing (BD) (Perić [109]) represents an attempt to preserve both boundedness and accuracy of the solution. It is a linear combination of UD, Eqn. (3.21) and CD, Eqn. (3.19) :

$$\phi_f = (1 - \gamma) \phi_f)_{UD} + \gamma \phi_f)_{CD}, \quad (3.22)$$

or

$$\begin{aligned} \phi_f = & [(1 - \gamma) \max(\text{sgn } F), 0] \phi_P \\ & + [(1 - \gamma) \min(\text{sgn}(F), 0) + \gamma(1 - f_x)] \phi_N. \end{aligned} \quad (3.23)$$

The blending factor γ , $0 \leq \gamma \leq 1$, determines how much numerical diffusion will be introduced. Perić [109] proposes a constant γ for all faces of the mesh. For $\gamma = 0$ the scheme reduces to UD.

Many other attempts to find an acceptable compromise between accuracy and boundedness have been made (see Section 1.2.1). The most promising approach at this stage combines a higher-order scheme with Upwind Differencing on a face-by-face basis, based on different boundedness criteria. We shall return to the issues of boundedness, accuracy and convergence of convection differencing schemes in Section 3.4.

3.3.1.3 Diffusion Term

The diffusion term will be discretised in a similar way. Using the assumption of linear variation of ϕ and Eqn. (3.15), it follows:

$$\begin{aligned} \int_{V_P} \nabla \cdot (\rho \Gamma_\phi \nabla \phi) dV &= \sum_f \mathbf{S} \cdot (\rho \Gamma_\phi \nabla \phi)_f \\ &= \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f. \end{aligned} \quad (3.24)$$

If the mesh is orthogonal, *i.e.* vectors $\mathbf{d}$ and $\mathbf{S}$ in Fig. 3.3 are parallel, it is possible

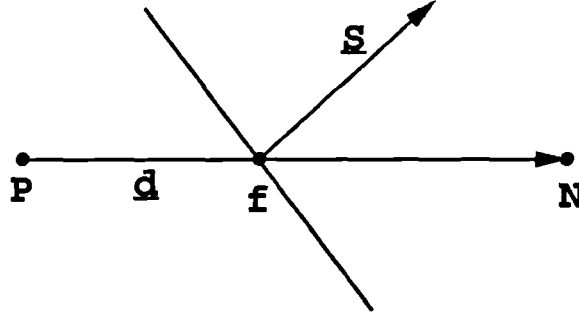


Figure 3.3: Vectors $\mathbf{d}$ and $\mathbf{S}$ on a non-orthogonal mesh.

to use the following expression:

$$\mathbf{S} \cdot (\nabla \phi)_f = |\mathbf{S}| \frac{\phi_N - \phi_P}{d}. \quad (3.25)$$

Using Eqn. (3.25), the face gradient of ϕ can be calculated from the two values around the face. An alternative would be to calculate the cell-centred gradient for the two cells sharing the face as:

$$(\nabla \phi)_P = \frac{1}{V_P} \sum_f \mathbf{S} \phi_f, \quad (3.26)$$

interpolate it to the face:

$$(\nabla\phi)_f = f_x(\nabla\phi)_P + (1 - f_x)(\nabla\phi)_N \quad (3.27)$$

and dot it with $\mathbf{S}$. Although both of the above-described methods are second-order accurate, Eqn. (3.27) uses a larger computational molecule. The first term of the truncation error is now four times larger than in the first method, which in turn cannot be used on non-orthogonal meshes.

Unfortunately, mesh orthogonality is more an exception than a rule. In order to make use of the higher accuracy of Eqn. (3.25), the product $\mathbf{S} \cdot (\nabla\phi)_f$ is split into two parts:

$$\mathbf{S} \cdot (\nabla\phi)_f = \underbrace{\Delta \cdot (\nabla\phi)_f}_{\text{orthogonal contribution}} + \underbrace{\mathbf{k} \cdot (\nabla\phi)_f}_{\text{non-orthogonal correction}} \quad (3.28)$$

The two vectors introduced in Eqn. (3.28), Δ and $\mathbf{k}$, have got to satisfy the following condition:

$$\mathbf{S} = \Delta + \mathbf{k}. \quad (3.29)$$

Vector Δ is chosen to be parallel with $\mathbf{d}$. This allows us to use Eqn. (3.25) on the orthogonal contribution, limiting the less accurate method only to the non-orthogonal part which cannot be treated in any other way.

Many possible decompositions exist and we will examine three:

- **Minimum correction approach.** The decomposition of $\mathbf{S}$, Fig. 3.4, is done

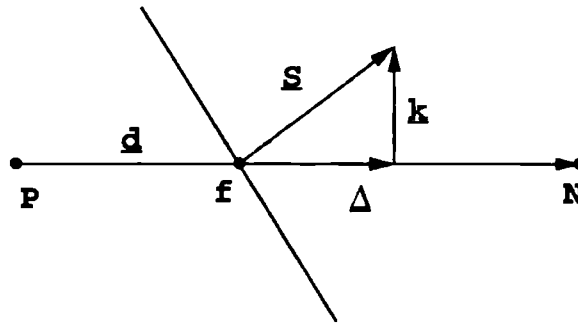


Figure 3.4: Non-orthogonality treatment with the “minimum correction” approach

in such a way to keep the non-orthogonal correction in Eqn. (3.28) as small as possible, by making Δ and $\mathbf{k}$ orthogonal:

$$\Delta = \frac{\mathbf{d} \cdot \mathbf{S}}{\mathbf{d} \cdot \mathbf{d}} \mathbf{d}, \quad (3.30)$$

with $\mathbf{k}$ calculated from Eqn. (3.29). As the non-orthogonality increases, the contribution from ϕ_P and ϕ_N decreases.

- **Orthogonal correction approach.** This approach keeps the contribution

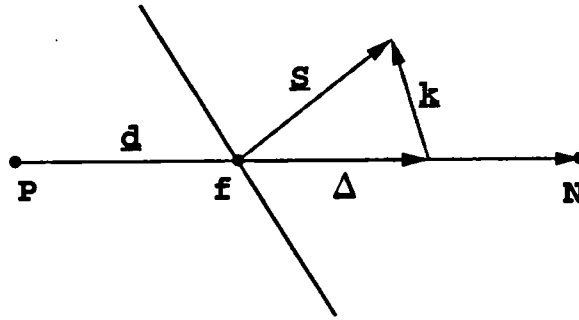


Figure 3.5: Non-orthogonality treatment in the “orthogonal correction” approach.

from ϕ_P and ϕ_N the same as on the orthogonal mesh irrespective of the non-orthogonality, Fig. 3.5. To achieve this we define:

$$\Delta = \frac{\mathbf{d}}{|\mathbf{d}|} |\mathbf{S}|. \quad (3.31)$$

- **Over-relaxed approach.** In this approach, the importance of the term in

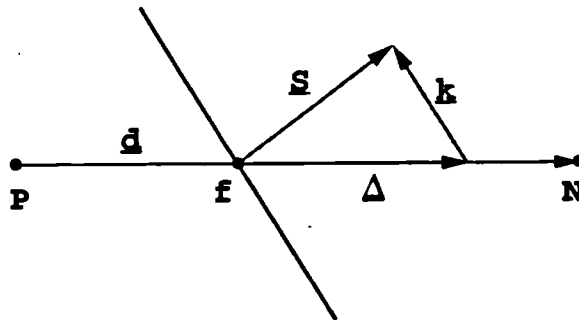


Figure 3.6: Non-orthogonality treatment in the “over-relaxed” approach.

ϕ_P and ϕ_N is caused to increase with the increase in non-orthogonality:

$$\Delta = \frac{\mathbf{d}}{\mathbf{d} \cdot \mathbf{S}} |\mathbf{S}|^2. \quad (3.32)$$

The decomposition of the face area vector is shown in Fig. 3.6.

The diffusion term, Eqn. (3.24), in its differential form exhibits the bounded behaviour. Its discretised form will preserve this property only on orthogonal meshes. The non-orthogonal correction potentially creates unboundedness, particularly if mesh non-orthogonality is high. If the preservation of boundedness is more important than accuracy, the non-orthogonal correction has got to be limited or completely discarded, thus violating the order of accuracy of the discretisation. This will be further discussed in Section 3.6.

All of the approaches described above are valid. Eqn. (3.29) is satisfied for all of them. The difference occurs in their accuracy and stability on non-orthogonal meshes and will be addressed later, namely in Sections 3.6 and 3.7.4.

The final form of the discretised diffusion term is the same for all three approaches. The orthogonal part of Eqn. (3.28) is discretised in the following way: since $\mathbf{d}$ and Δ are parallel, it follows that:

$$\Delta \cdot (\nabla \phi)_f = |\Delta| \frac{\phi_N - \phi_P}{\mathbf{d}} \quad (3.33)$$

and Eqn. (3.28) can be written as:

$$\mathbf{S} \cdot (\nabla \phi)_f = |\Delta| \frac{\phi_N - \phi_P}{\mathbf{d}} + \mathbf{k} \cdot (\nabla \phi)_f. \quad (3.34)$$

The face interpolate of $\nabla \phi$ is calculated using Eqn. (3.27).

3.3.1.4 Source Terms

All terms of the original equation that cannot be written as convection, diffusion or temporal terms are treated as sources. The source term, $S_\phi(\phi)$, can be a general function of ϕ . When deciding on the form of the discretisation for the source, its interaction with other terms in the equation and its influence on boundedness and accuracy should be examined. Some general comments on the treatment of source terms are given in Patankar [105]. A simple procedure will be explained here.

Before the actual discretisation, the source term needs to be linearised:

$$S_\phi \phi = Su + Sp\phi, \quad 3.35$$

where Su and Sp can also depend on ϕ . Following Eqn. (3.12), the volume integral is calculated as:

$$\int_{V_P} S_\phi(\phi) dV = Su V_P + Sp V_P \phi_P. \quad (3.36)$$

The importance of the linearisation becomes clear in implicit calculations. It is advisable to treat the source term as “implicitly” as possible. This will be further explained in Section 3.5.

3.3.2 Temporal Discretisation

In the previous Section, the discretisation of spatial terms has been presented. This can be split into two parts – the transformation of surface and volume integrals into discrete sums and expressions that give the face values of the variable as a function of cell values. Let us again consider the integral form of the transport equation, Eqn. (3.8):

$$\begin{aligned} \int_t^{t+\Delta t} \left[\frac{\partial}{\partial t} \int_{V_P} \rho \phi dV + \int_{V_P} \nabla \cdot (\rho \mathbf{U} \phi) dV - \int_{V_P} \nabla \cdot (\rho \Gamma_\phi \nabla \phi) dV \right] dt \\ = \int_t^{t+\Delta t} \left(\int_{V_P} S_\phi(\phi) dV \right) dt. \end{aligned}$$

Using Eqs. (3.17), (3.34) and (3.36), and assuming that the control volumes do not change in time, Eqn. (3.8) can be written as:

$$\begin{aligned} \int_t^{t+\Delta t} \left[\left(\frac{\partial \rho \phi}{\partial t} \right)_P V_P + \sum_f F \phi_f - \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f \right] dt \\ = \int_t^{t+\Delta t} (Su V_P + Sp V_P \phi_P) dt. \end{aligned} \quad (3.37)$$

The above expression is usually called the “semi-discretised” form of the transport equation (Hirsch [65]).

Having in mind the prescribed variation of the function in time, Eqn. (3.4), the temporal integrals and the time derivative can be calculated directly as:

$$\left(\frac{\partial \rho \phi}{\partial t} \right)_P = \frac{\rho_P^n \phi_P^n - \rho_P^o \phi_P^o}{\Delta t}, \quad (3.38)$$

$$\int_t^{t+\Delta t} \phi dt = \frac{1}{2} (\phi^o + \phi^n) \Delta t, \quad (3.39)$$

where

$$\begin{aligned}\phi^n &= \phi(t + \Delta t), \\ \phi^o &= \phi(t).\end{aligned}\tag{3.40}$$

Assuming that the density and diffusivity do not change in time, Eqs. (3.37), (3.38) and (3.39) give:

$$\begin{aligned}\frac{\rho_P \phi_P^n - \rho_P \phi_P^o}{\Delta t} V_P &+ \frac{1}{2} \sum_f F \phi_f^n - \frac{1}{2} \sum_f \rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f^n \\ &+ \frac{1}{2} \sum_f F \phi_f^o - \frac{1}{2} \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f^o \\ &- Su V_P + \frac{1}{2} Sp V_P \phi_P^n + \frac{1}{2} Sp V_P \phi_P^o.\end{aligned}\tag{3.41}$$

This form of temporal discretisation is called the **Crank-Nicholson** method. It is second-order accurate in time. It requires the face values of ϕ and $\nabla \phi$ as well as the cell values for both old and new time-level. The face values are calculated from the cell values on each side of the face, using the appropriate differencing scheme for the convection term, and Eqn. (3.34) for diffusion. The evaluation of the non-orthogonal correction term will be discussed later (see Section 3.5). Our task is to determine the new value of ϕ_P . Since ϕ_f and $(\nabla \phi)_f$ also depend on values of ϕ in the surrounding cells, Eqn. (3.41) produces an algebraic equation:

$$a_P \phi_P^n + \sum_N a_N \phi_N^n = R_P.\tag{3.42}$$

For every control volume, one equation of this form is assembled. The value of ϕ_P^n depends on the values in the neighbouring cells, thus creating a system of algebraic equations:

$$[A] [\phi] = [R],\tag{3.43}$$

where $[A]$ is a sparse matrix, with coefficients a_P on the diagonal and a_N off the diagonal, $[\phi]$ is the vector of ϕ -s for all control volumes and $[R]$ is the source term vector. The sparseness pattern of the matrix depends on the order in which the control volumes are labelled, with every off-diagonal coefficient above and below the

diagonal corresponding to one of the faces in the mesh. In the rest of this study, Eqn. (3.43) will be represented by the typical equation for the control volume, Eqn. (3.42).

When this system is solved, it gives a new set of ϕ values – the solution for the new time-step. As will be shown later, the coefficient a_P in Eqn. (3.42) includes the contribution from all terms corresponding to ϕ_P^n – the temporal derivative, convection and diffusion terms as well as the linear part of the source term. The coefficients a_N include the corresponding terms for each of the neighbouring points. The summation is done over all CV-s that share a face with the current control volume. The source term includes all terms that can be evaluated without knowing the new ϕ 's, namely, the constant part of the source term, and the parts of the temporal derivative, convection and diffusion terms corresponding to the old time-level.

The Crank-Nicholson method of temporal discretisation is unconditionally stable (Hirsch [65]), but does not guarantee boundedness of the solution. Examples of unrealistic solutions given by the Crank-Nicholson scheme can be found in Patankar and Baliga [106]. As in the case of the convection term, boundedness can be obtained if the equation is discretised to first order temporal accuracy.

It has been customary to neglect the variation of the face values of ϕ and $\nabla\phi$ in time (Patanekar [105]). This leads to several methods of temporal discretisation. The new form of the discretised transport equation combines the old and new time-level convection, diffusion and source terms, leaving the temporal derivative unchanged:

$$\begin{aligned} \frac{\rho_P \phi_P^n - \rho_P \phi_P^o}{\Delta t} V_P + \sum_f F \phi_f - \sum_f (\rho \Gamma_\phi)_f S.(\nabla \phi)_f \\ = SuV_P + SpV_P \phi_P. \end{aligned} \quad (3.44)$$

The resulting equation is only first-order accurate in time and a choice has to be made about the way the face values of ϕ and $\nabla\phi$ are evaluated.

In **explicit discretisation**, the face values of ϕ and $\nabla\phi$ are determined from

the old time-field:

$$\phi_f = f_x \phi_P^o + (1 - f_x) \phi_N^o, \quad (3.45)$$

$$\mathbf{S} \cdot (\nabla \phi)_f = \Delta \frac{\phi_N^o - \phi_P^o}{\mathbf{d}} + \mathbf{k} \cdot (\nabla \phi)_f^o. \quad (3.46)$$

The linear part of the source term is also evaluated using the old-time value. Eqn. (3.44) can be written in the form:

$$\phi_P^n = \phi_P^o + \frac{\Delta t}{\rho_P V_P} \left[\sum_f F \phi_f - \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f + S_u V_P + S_p V_P \phi_P^o \right]. \quad (3.47)$$

The consequence of this choice is that all terms on the r.h.s. of Eqn. (3.47) depend only on the old-time field. The new value of ϕ_P can be calculated directly – it is no longer necessary to solve the system of linear equations. The drawback of this method is the Courant number limit (Courant *et al.* [32]). The Courant number is defined as:

$$Co = \frac{\mathbf{U}_f \cdot \mathbf{d}}{\Delta t}. \quad (3.48)$$

If the Courant number is larger than unity, the explicit system becomes unstable. This is a severe limitation, especially if we are trying to solve a steady-state problem.

The **Euler Implicit method** expresses the face-values in terms of the new time-level cell values²:

$$\phi_f = f_x \phi_P^n + (1 - f_x) \phi_N^n, \quad (3.49)$$

$$\mathbf{S} \cdot (\nabla \phi)_f = \Delta \frac{\phi_N^n - \phi_P^n}{\mathbf{d}} + \mathbf{k} \cdot (\nabla \phi)_f. \quad (3.50)$$

This is still only first order accurate but, unlike the explicit method, it creates an system of equations like Eqn. (3.42). The coupling in the system is much stronger than in the explicit approach and the system is stable even if the Courant number limit is violated (Hirsch [65]). Unlike the explicit method, this form of temporal discretisation guarantees boundedness

²The evaluation of the non-orthogonal correction will be discussed later in Section 3.5

Backward Differencing in time is a temporal scheme which is second-order accurate in time and still neglects the temporal variation of the face values. In order to achieve this, each individual term of Eqn. (3.8) needs to be discretised to second order accuracy.

The discretised form of the temporal derivative in Eqn. (3.44) can be obtained in the following way: consider the Taylor series expansion of ϕ in time around $\phi(t + \Delta t) = \phi^n$:

$$\phi(t) = \phi^o = \phi^n - \frac{\partial \phi}{\partial t} \Delta t + \frac{1}{2} \frac{\partial^2 \phi}{\partial t^2} \Delta t^2 + O(\Delta t^3). \quad (3.51)$$

The temporal derivative can therefore be expressed as:

$$\frac{\partial \phi}{\partial t} = \frac{\phi^n - \phi^o}{\Delta t} + \frac{1}{2} \frac{\partial^2 \phi}{\partial t^2} \Delta t + O(\Delta t^2). \quad (3.52)$$

In spite of the prescribed linear variation of ϕ in time, Eqn. (3.38) approximates the temporal derivative only to first order accuracy, since the first term of the truncation error in Eqn. (3.52) scales with Δt . However, if the temporal derivative is discretised up to second order, the whole discretisation of the transport equation will be second-order accurate even if the temporal variation of ϕ_f and $(\nabla \phi)_f$ is neglected.

In order to achieve this, the Backward Differencing in time uses three time levels. The additional Taylor series expansion for the “second old” time level is:

$$\phi(t - \Delta t) = \phi^{oo} = \phi^n - 2 \frac{\partial \phi}{\partial t} \Delta t + 2 \frac{\partial^2 \phi}{\partial t^2} \Delta t^2 + O(\Delta t^3). \quad (3.53)$$

It is now possible to eliminate the term in the truncation error which scales with Δt . Combining Eqs. (3.51) and (3.53) the second-order approximation of the temporal derivative is:

$$\frac{\partial \phi}{\partial t} = \frac{\frac{3}{2} \phi^n - 2 \phi^o + \frac{1}{2} \phi^{oo}}{\Delta t}. \quad (3.54)$$

Again, the boundedness of the solution cannot be guaranteed. Comparison between the Backward Differencing and the Crank-Nicholson method will be made in Section 3.6. The final form of the discretised equation with Backward Differencing in

time is:

$$\begin{aligned} \frac{\frac{3}{2}\rho_P\phi^n - 2\rho_P\phi^o + \frac{1}{2}\rho_P\phi^{oo}}{\Delta t}V_P + \sum_f F\phi_f^n - \sum_f (\rho\Gamma_\phi)_f^n \mathbf{S} \cdot (\nabla\phi)_f \\ = SuV_P + SpV_P\phi_P^n. \end{aligned} \quad (3.55)$$

This produces a system of algebraic equations that must be solved for ϕ_P^n .

Steady-state problems are quite common in fluid flows. Their characteristic is that the solution is not a function of time, *i.e.* the transport equation reduces to:

$$\nabla \cdot (\rho \mathbf{U} \phi) - \nabla \cdot (\rho \Gamma_\phi \nabla \phi) = Su + Sp\phi. \quad (3.56)$$

If we are solving a single equation of this type, the solution can be obtained in a single step. This is generally not the case: fluid flow problems require a solution of non-linear systems of coupled equations. If the non-linearity of the system is lagged, which is the case in the segregated approach used in this study, it is still necessary to solve the system in an iterative manner. In order to speed up the convergence, an implicit formulation is preferred. The convergence of the iterative procedure can be improved through under-relaxation, which will be described in Section 3.5.

3.3.3 Implementation of Boundary Conditions

Let us now consider the implementation of boundary conditions. The computational mesh includes a series of faces which coincide with the boundaries of the physical domain under consideration. The conditions there are prescribed through the boundary conditions.

In order to simplify the discussion, the boundary conditions are divided into **numerical** and **physical boundary conditions**.

There are two basic types of numerical boundary conditions. Dirichlet (or fixed value) boundary condition prescribes the value of the variable on the boundary. Von Neumann boundary condition, on the other hand, prescribes the gradient of the variable normal to the boundary. These two types of boundary conditions can be built into the system of algebraic equations, Eqn. (3.42), before the solution.

Physical boundary conditions are symmetry planes, walls, inlet and outlet conditions for fluid flow problems, adiabatic or fixed temperature boundaries for heat transfer problems *etc.*. Each of these conditions is associated with a set of numerical boundary conditions on each of the variables that is being calculated. Some more complicated boundary conditions (radiation boundaries, for example) may specify the interaction between the boundary value and the gradient on the boundary.

3.3.3.1 Numerical Boundary Conditions

Before we consider the implementation of numerical boundary conditions, we have to address the treatment of non-orthogonality on the boundary. Consider a control volume with a boundary face b , shown in Fig. 3.7. In this situation, the vector $\mathbf{d}$

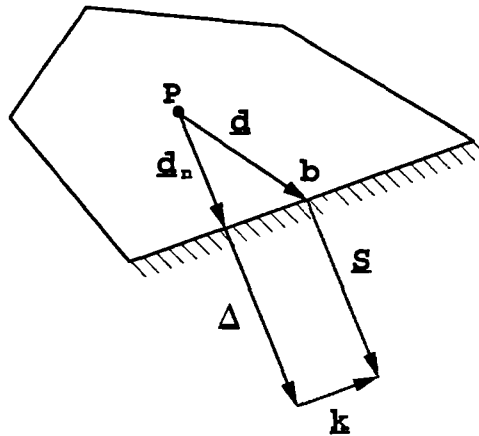


Figure 3.7: Control volume with a boundary face.

extends only to the centre of the boundary face.

It is assumed that a boundary condition specified for the boundary face is valid along the whole of the face. The decomposition of the face area vector into the orthogonal and non-orthogonal part is now simple: the vector $\mathbf{k}$ in Fig. 3.7 is parallel to the face. The orthogonal part of the face area vector (Δ in Fig. 3.7) is therefore equal to $\mathbf{S}$, but is no longer located in the middle of the face.

The vector between the cell centre and the boundary face is now normal to the

boundary:

$$\mathbf{d}_n = \frac{\mathbf{S}}{S} \frac{d \cdot \mathbf{S}}{S}, \quad (3.57)$$

and the correction vector $\mathbf{k}$ is not used.

- **Fixed Value Boundary Condition**

The fixed value boundary condition prescribes the value of ϕ at the face b to be ϕ_b . This has to be taken into account in the discretisation of the convection and diffusion terms on the boundary face.

- **Convection term.** According to Eqn. (3.17), the convection term is discretised as:

$$\int_{V_P} \nabla \cdot (\rho \mathbf{U} \phi) dV = \sum_f F \phi_f.$$

It is known that the value of ϕ on the boundary face is ϕ_b . Therefore, the term for the boundary face is:

$$F_b \phi_b, \quad (3.58)$$

where F_b is the face flux.

- **Diffusion term.** The diffusion term is discretised according to Eqn. (3.24).

$$\int_{V_P} \nabla \cdot (\rho \Gamma_\phi \nabla \phi) dV = \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f.$$

The face gradient at b is calculated from the known face value and the cell centre value:

$$\mathbf{S} \cdot (\nabla \phi)_b = S \frac{\phi_b - \phi_P}{|\mathbf{d}_n|}, \quad (3.59)$$

because $\mathbf{S}$ and $\mathbf{d}_n$ are parallel.

- **Fixed Gradient Boundary Condition**

In the case of the fixed gradient boundary condition, the dot-product of the gradient and the outward pointing unit normal is prescribed on the boundary:

$$\left(\frac{\mathbf{S}}{S} \cdot \nabla \phi \right)_b = g_b. \quad (3.60)$$

- **Convection term.** The face value of ϕ is calculated from the value in the cell centre and the prescribed gradient:

$$\begin{aligned}\phi_b &= \phi_P + \mathbf{d}_n \cdot (\nabla \phi)_b \\ &= \phi_P + \mathbf{d}_n \cdot \mathbf{g}_b.\end{aligned}\tag{3.61}$$

- **Diffusion term.** The dot product between the face area vector and $(\nabla \phi)_b$ is known to be

$$\mathbf{S} \cdot \mathbf{g}_b\tag{3.62}$$

and the resulting term is:

$$(\rho \Gamma_\phi)_b \mathbf{S} \cdot \mathbf{g}_b.\tag{3.63}$$

As the vector $\mathbf{d}_n$ does not point to the middle of the boundary face, the face integrals in the fixed gradient boundary condition are calculated only to first order accuracy. This can be rectified by including the boundary face correction based on the vector $\mathbf{k}$ (Fig. 3.7) and the component of the gradient parallel to the face in the first cell next to the boundary. However, Section 3.6 will show that the error of the same type is neglected for the internal faces of the mesh and this correction is omitted for the sake of consistency.

3.3.3.2 Physical Boundary Conditions

Let us now consider some physical boundary conditions for fluid flow calculations. For simplicity, we shall start with the **incompressible flow**.

- **Inlet boundary.** The velocity field at the inlet is prescribed and, for consistency, the boundary condition on pressure is zero gradient (Hirsch [65]).
- **Outlet boundary.** The outlet boundary condition should be specified in such a way that the overall mass balance for the computational domain is satisfied. This can be done in two ways:

- The velocity distribution for the boundary is projected from the inside of the domain (first row of cells next to the boundary). These velocities are scaled to satisfy overall continuity. This approach, however, leads to instability if inflow through a boundary specified as outlet occurs. The boundary condition on pressure is again zero gradient.
- The other approach does not require the velocity distribution across the outlet – the pressure distribution is specified instead. The fixed value boundary condition is used for the pressure, with the zero gradient boundary condition on velocity. Overall mass conservation is guaranteed by the pressure equation (see Section 3.8).
- **Symmetry plane boundary.** The symmetry plane boundary condition implies that the component of the gradient normal to the boundary should be fixed to zero. The components parallel to it are projected to the boundary face from the inside of the domain.
- **Impermeable no-slip walls.** The velocity of the fluid on the wall is equal to that of the wall itself, so the fixed value boundary conditions prevail. As the flux through the solid wall is known to be zero, the pressure gradient condition is zero gradient.

Compressible flows at low Mach numbers are subject to the same approach as above. The situation is somewhat more complex in case of transonic and supersonic flows – the number of boundary conditions fixed at the inlet and outlet depends on the number of characteristics pointing into the domain. For these cases the reader is referred to Hirsch [65] or Uslu [135].

For **turbulent flows**, the inlet and outlet boundary conditions on turbulence variables (k and ϵ , for example) are typically assigned to fixed values and zero gradients, respectively. The boundary conditions for the turbulence properties on the wall depend on the form of the selected turbulence model and the near-wall treatment.

3.4 A New Convection Differencing Scheme

In this Section, a new convection differencing scheme will be proposed. It is based on the Normalised Variable Approach (NVA) of Leonard [82] and Gaskell and Lau [49]. It guarantees boundedness of the solution by introducing a small amount of numerical diffusion in regions where it is needed. The scheme is formulated as a stabilised version of Central Differencing (CD) in order to preserve overall second-order accuracy. The necessary modification of the Convection Boundedness Criterion of Gaskell and Lau [49] for non-uniform unstructured meshes is also given.

The description of the new differencing scheme will start with an analysis of the issues of accuracy and boundedness, and introduction of the concepts of Total Variation Diminishing (TVD) and the Normalised Variable Approach (NVA).

3.4.1 Accuracy and Boundedness

As has been mentioned before, none of the basic convection differencing schemes produces a numerical solution which is bounded and accurate at the same time. This problem has been tackled in several different ways, including the introduction of “streamwise-upwind schemes” (*eg.* Raithby [112, 113]), higher-order upwind schemes (*eg.* QUICK by Leonard [81]) and flux-limited schemes (*eg.* FCT by Boris and Book [20]). Flux limiting is a procedure that creates a differencing scheme which is higher than first-order accurate, but without the spurious oscillations associated with the classical second-order schemes. This has proven to be the most promising approach. Harten [58] introduces the notion of Total Variation Diminishing (TVD) to characterise oscillation-free flux-limited schemes.

3.4.1.1 TVD Differencing Schemes

The derivation of the TVD criterion starts with the derivation of semi-discrete schemes for the unsteady convection problems by Osher, named the “*E*-schemes”. He shows that these schemes are at most first-order accurate but converge to the

correct physical solution which satisfies the entropy condition³.

It is well known (*eg.* Sweby [129], Harten [58]) that a crucial estimate involved in convergence proofs of differencing schemes is a bound on the variation of the solution. The Total Variation $TV(\phi^{n+1})$ [129] of the solution is defined by:

$$TV(\phi^n) = \sum_f \phi_N^n - \phi_P^n, \quad (3.64)$$

where P and N are the points around the face f . The Total Variation Diminishing schemes satisfy the following condition (Sweby [129]) for every time-step:

$$TV(\phi^{n+1}) \leq TV(\phi^n). \quad (3.65)$$

Sweby also shows that the schemes satisfying the above condition are also “ E -schemes”, thus producing the solution that obeys the entropy condition.

The above condition is now applied to the higher-order flux-limited schemes in the way suggested by Sweby [129]: a selected higher-order differencing scheme is written as a sum of the first-order bounded differencing scheme (UD) and a “limited” higher order correction:

$$\phi_f = (\phi)_{UD} + \Psi[(\phi)_{HO} - (\phi)_{UD}], \quad (3.66)$$

where $(\phi)_{HO}$ represents the face value of ϕ for the selected higher-order scheme and Ψ is the flux limiter. Following van Leer [139] and Chakravarthy and Osher [27], Sweby [129] assumes that the limiter is a function of consecutive gradients of ϕ , Fig 3.8, as measured by:

$$r = \frac{\phi_C - \phi_U}{\phi_D - \phi_C}. \quad (3.67)$$

Thus:

$$\Psi = \Psi(r). \quad (3.68)$$

The points U , C and D are selected according to the direction of the flow on the face f .

³ The weak solutions of this equation are non-unique (Sweby [129]) and so an extra constraint is needed to select the unique physical solution.

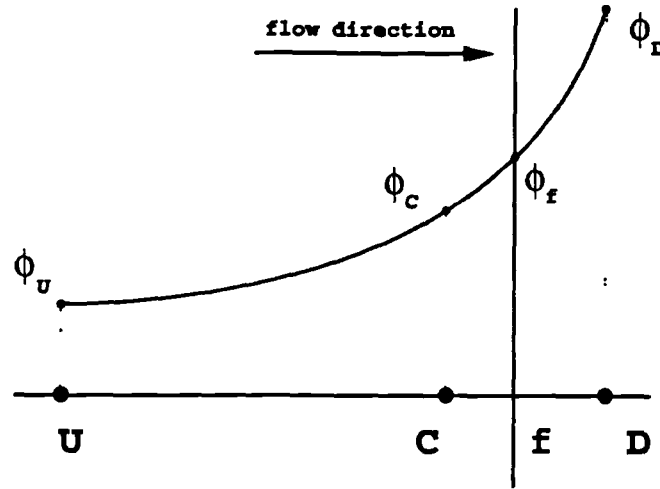


Figure 3.8: Variation of ϕ around the face f .

Sweby [129] shows that the following constraint on the limiter guarantees that the scheme satisfies the TVD condition:

$$0 \leq \left(\frac{\Psi(r)}{r}, \Psi(r) \right) \leq 2. \quad (3.69)$$

The points that satisfy the TVD condition can be presented as the shaded region in Sweby's diagram, Fig. 3.9, showing Ψ as a function of r . The hatched region in Fig. 3.9 is known as the second-order region of the TVD diagram (Sweby [129]).

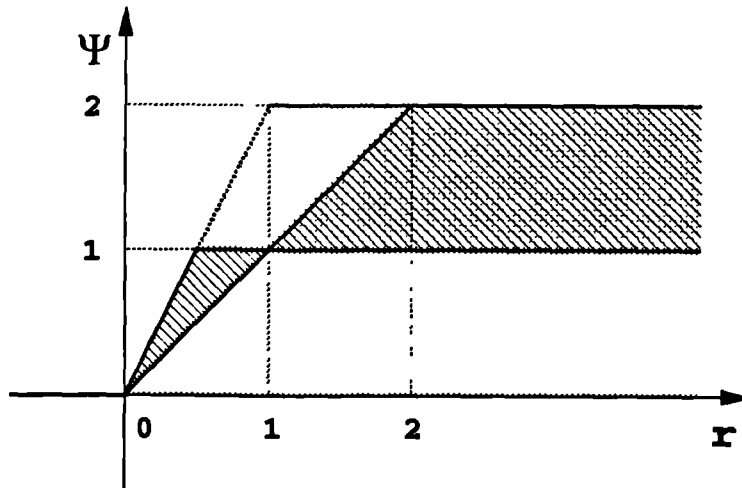


Figure 3.9: Sweby's diagram.

The TVD boundedness criterion as derived by Sweby [129] includes the dependence of the limited scheme on the Courant number. The TVD condition given

by Eqn. (3.69), applies to explicit second-order differencing schemes on a five-point computational molecule in one-dimensional problems. All second-order schemes are considered to be a linear combination of explicit versions of Linear Upwinding (Warming and Beam [146]) and Central Differencing (Lax-Wendroff [79]), where the face value of ϕ depends on the Courant number in order to guarantee stability. As the Courant number increases, the differencing scheme becomes more and more diffusive and in the limit of $Co = 1$, both schemes reduce to Upwind Differencing. This type of behaviour is essential to prove the convergence of TVD differencing schemes.

The dependence of the convection differencing scheme on the Courant number cannot be accepted – it would imply that a steady-state solution for a convection-dominated problem depends on the time-step used to reach the steady state.

In spite of its deficiencies, the TVD approach has been widely accepted. It has, however, been noted (Leonard [83]) that TVD schemes are either too diffusive, introducing too much numerical diffusion (for example MINMOD by Zhu and Rodi [155]), or too compressive, distorting the smoothly changing solutions through the introduction of “down-winding”, where the influence of ϕ_D to ϕ_f is larger than that of ϕ_C (SUPERBEE by Roe [118]).

A less restrictive and more general approach to boundedness can be found in the Normalised Variable Approach (NVA), described below.

3.4.1.2 Convection Boundedness Criterion and the NVD Diagram

The Normalised Variable Approach and Convection Boundedness Criterion have been introduced by Leonard [82] and Gaskell and Lau [49]. Unlike the TVD approach, it does not guarantee convergence of the proposed differencing scheme, but tackles the problem of boundedness in a more general way.

Consider the variation of a convected scalar ϕ around the face f in a one dimensional case, Fig. 3.8. The normalised variable is defined as [82]

$$\tilde{\phi} = \frac{\phi - \phi_U}{\phi_D - \phi_U}, \quad 3.70$$

where U and D refer to the points in Fig. 3.8.

With this definition, any differencing scheme using only the nodal values of ϕ at points U , C and D to evaluate the face value ϕ_f can be written in the form:

$$\tilde{\phi}_f = f(\tilde{\phi}_C), \quad (3.71)$$

where

$$\tilde{\phi}_C = \frac{\phi_C - \phi_U}{\phi_D - \phi_U} \quad (3.72)$$

and

$$\tilde{\phi}_f = \frac{\phi_f - \phi_U}{\phi_D - \phi_U}. \quad (3.73)$$

In order to avoid unphysical oscillations in the solution, we will require that ϕ_C (and consequently ϕ_f) are locally bounded between ϕ_U and ϕ_D , meaning either

$$\phi_U \leq \phi_C \leq \phi_D, \quad (3.74)$$

or

$$\phi_U \geq \phi_C \geq \phi_D. \quad (3.75)$$

If this criterion is satisfied for every point in the domain, the entire solution will be free of any unphysical oscillations. This is the basis of the Normalised Variable Approach.

The Convection Boundedness Criterion (CBC) (Gaskell and Lau [49]) for ϕ_C is given as:

$$0 \leq \tilde{\phi}_C \leq 1, \quad (3.76)$$

because by definition

$$\tilde{\phi}_C = 0 \Rightarrow \phi_C = \phi_U, \quad (3.77)$$

$$\tilde{\phi}_C = 1 \Rightarrow \phi_C = \phi_D. \quad (3.78)$$

Gaskell and Lau [49] show that the boundedness criterion for convection differencing schemes can be presented in the Normalised Variable Diagram (NVD), showing $\tilde{\phi}_f$ as a function of $\tilde{\phi}_C$, as the hatched region in Fig. 3.10 (including the line $\tilde{\phi}_f = \tilde{\phi}_C$), or with the following conditions:

- for $0 \leq \tilde{\phi}_C < 1$, $\tilde{\phi}_f$ is bounded below by the function $\phi_f = \tilde{\phi}_C$ and above by unity, and passes through the points $(0,0)$ and $(1,1)$,
- for $\tilde{\phi}_C < 0$ and $\tilde{\phi}_C > 1$ $\tilde{\phi}_f$ is equal to $\tilde{\phi}_C$.

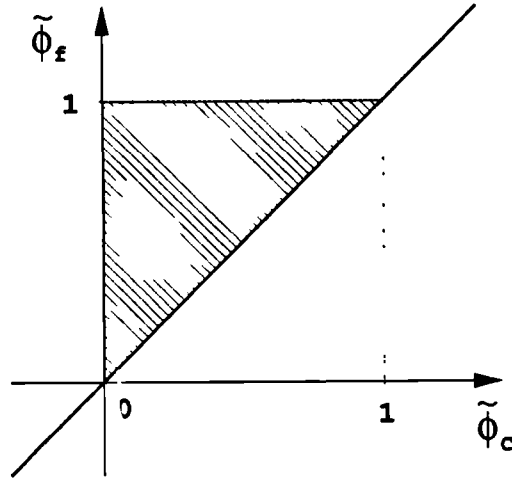


Figure 3.10: Convection Boundedness Criterion in the Normalised Variable Diagram.

Fig. 3.11 shows some basic differencing schemes represented in the NVD: Upwind Differencing (UD), Central Differencing (CD), Linear Upwind Differencing (LUD) by Warming and Beam [146] and QUICK by Leonard [81]. It can be seen that the only basic differencing scheme that satisfies the boundedness criterion is UD.

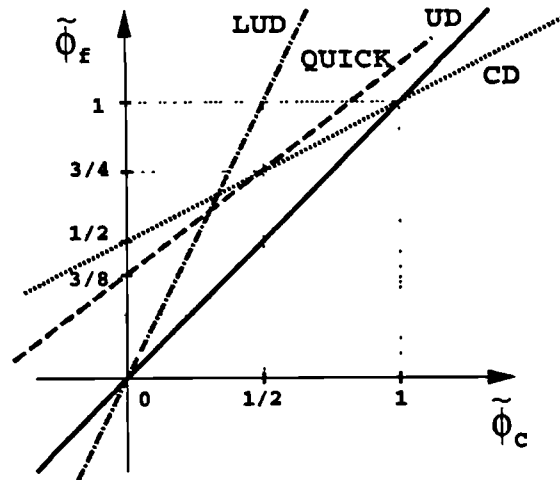


Figure 3.11: Common differencing schemes in the NVD diagram

It should be noted that the NVD makes no reference to the Courant number, thus resolving the problem of dependence of the steady-state solution on the time-step. In a later paper, Leonard [83] includes the “transient interpolation modelling” of the face value of ϕ depending on the time-step, again restricting all differencing schemes to UD for $Co = 1$.

NVD diagrams for a number of TVD and NVD schemes as well as the relationship between NVD and Sweby’s diagram can be found in Leonard [83].

3.4.1.3 Convergence Problems of Flux-Limited Schemes

According to both TVD and NVA, any differencing scheme that is more than first-order accurate must be non-linear in order to guarantee boundedness. Non-linearity of the differencing scheme is introduced through the dependence of the flux limiter on the solution itself (see Eqs. (3.68) and (3.71)).

This additional non-linearity does not cause the divergent behaviour, as the original higher-order scheme satisfies the stability condition. It has, however, been noted that the NVD schemes under some conditions do not produce a unique solution for a steady-state problem, as the change in the discretisation practice changes the solution between the time-steps and vice versa. For the TVD schemes, the convergence of the non-linear system to a unique solution can be proven only for explicit calculations, introducing the dependence of the limiter on the Courant number. In the case of implicit calculations in two or three dimensions, or for $Co > 1$, the convergence properties of TVD schemes cannot be proven. The NVD criterion, on the other hand, does not guarantee that the differencing scheme will be convergent under any conditions. Experience with some NVD differencing schemes (Zhu [154]) shows that special modifications in the implementation of the differencing scheme are needed in order to improve convergence.

The creation of a differencing scheme in either the TVD or NVA framework can be divided into three steps:

- A basic differencing scheme needs to be selected. This scheme can be unbounded, but should be accurate and stable. This basic differencing scheme

will be used unaltered in those regions where it produces a bounded solution.

- The regions of the domain in which problems with boundedness occur need to be detected in some way. For this purpose the TVD schemes use the “smoothness monitor” r , Eqn. (3.67), whereas the NVD schemes introduce the concept of the “normalised variable” $\tilde{\phi}$, Eqn. (3.70).
- Finally, a change in the discretisation in regions of unboundedness needs to be introduced. This change is partially dictated by the TVD or NVD criterion itself, Figs. 3.9 and 3.10.

The Convection Boundedness Criterion represents the most general approach to the issue of local boundedness of the solution. It has therefore been selected as the boundedness indicator for the new bounded differencing scheme. According to its original formulators, the Normalised Variable Approach is not readily extendible to arbitrarily unstructured meshes, but in the present work we show that this is not the case. The next Section introduces the necessary changes in the calculation of the Normalised Variable to achieve this.

The choice of the basic differencing scheme comes from the desired order of accuracy of the Finite Volume method presented in this study. In order to achieve the second order behaviour, consistent with the rest of the discretisation practice, CD has been selected. It uses a compact computational molecule, which is advantageous for unstructured meshes.

The bounding process for the new differencing scheme is based on the NVD diagram. Once the basic framework has been established, the issue of accuracy *vs.* convergence will be addressed.

3.4.2 Modification of the NVD Criterion for Unstructured Meshes

The CBC uses the value of the dependent variable ϕ in the U far upwind node in order to determine $\tilde{\phi}_C$. This is not practical for arbitrarily unstructured meshes

because the far upwind cell label is not known, or that point does not even exist.

The definition of the normalised variable, Eqn. (3.70), is originally given for a uniform mesh. This implies that a differencing scheme based on Eqn. (3.71) will be sensitive to sudden changes in the mesh spacing. A generalisation of the NVD criterion to non-uniform meshes has been given by Darwish [37]. However, this generalisation is complex and introduces additional computational cost.

In this Section, a modification of the Normalised Variable in terms of gradients of the dependent variable will be presented. It enables the use of the NVA on arbitrarily unstructured and non-uniform meshes without an increase in the computational cost.

The information available for each cell face includes the gradients of the dependent variable over both cells sharing the face and the gradient across the face itself. Using the fact that the computational point is located in the middle of the control volume, definition of $\tilde{\phi}_C$ can be changed as follows.

If the value of ϕ_C is bounded by ϕ_U and ϕ_D , Fig. 3.12, it is also bounded by the interpolated face values ϕ_f^+ and ϕ_f^- . The advantage of this formulation is that the calculation of the gradient of ϕ across the cell on the basis of the interpolated face values is valid both on uniform and non-uniform meshes. The new definition of $\tilde{\phi}_C$

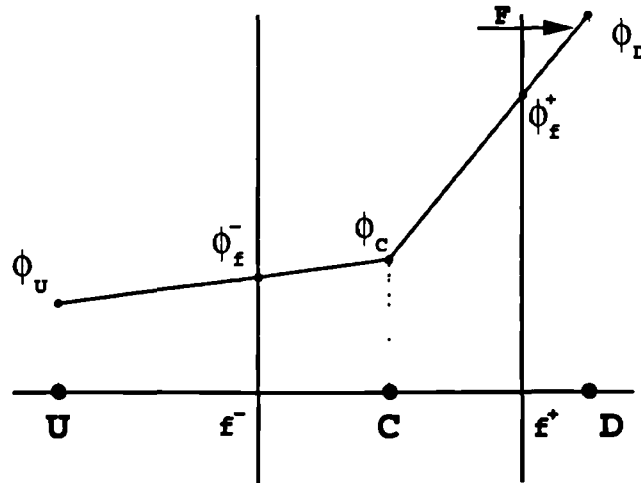


Figure 3.12: Modified definition of the boundedness criterion for unstructured meshes.

is therefore:

$$\tilde{\phi}_C = \frac{\phi_C - \phi_f^-}{\phi_f^+ - \phi_f^-} = 1 - \frac{\phi_f^+ - \phi_C}{\phi_f^+ - \phi_f^-}. \quad (3.79)$$

In the case of a uniform mesh, Eqn. (3.72) and Eqn. (3.79) are equivalent.

The next step is to reformulate Eqn. (3.79) in terms of gradients across the face and the “upwind” cell C . Using the interpolation factor f_x , Eqn. (3.20), and vector $\mathbf{d}$, Fig. 3.6, the difference $\phi_f^+ - \phi_C$ can be written as:

$$\begin{aligned}\phi_f^+ - \phi_C &= f_x(\phi_D - \phi_C) \\ &= f_x \frac{\phi_D - \phi_C}{x_D - x_C} (x_D - x_C) \\ &= f_x (\nabla \phi)_f \cdot \hat{\mathbf{d}} (x_D - x_C),\end{aligned}\quad (3.80)$$

since

$$\frac{\phi_D - \phi_C}{x_D - x_C} = (\nabla \phi)_f \cdot \hat{\mathbf{d}}. \quad (3.81)$$

Here, $\hat{\mathbf{d}}$ is the unit vector parallel with $\overline{CD}$:

$$\hat{\mathbf{d}} = \frac{\mathbf{d}}{d}. \quad (3.82)$$

The transformation of $\phi_f^+ - \phi_f$ can be done in the following way:

$$\begin{aligned}\phi_f^+ - \phi_f &= \frac{\phi_f^+ - \phi_f}{x_f^+ - x_f^-} (x_f^+ - x_f^-) \\ &= (\nabla \phi)_C \cdot \hat{\mathbf{d}} (x_f^+ - x_f^-).\end{aligned}\quad (3.83)$$

Using Eqs. 3.20, (3.80) and (3.83), the transformation of Eqn. (3.79) is simple:

$$\begin{aligned}\tilde{\phi}_C &= 1 - \frac{\phi_f^+ - \phi_C}{\phi_f^+ - \phi_f} \\ &= 1 - \frac{f_x (\nabla \phi)_f \cdot \hat{\mathbf{d}} (x_D - x_C)}{(\nabla \phi)_C \cdot \hat{\mathbf{d}} (x_f^+ - x_f^-)}.\end{aligned}\quad (3.84)$$

Since the computational point C is located in the middle of the cell, the following geometrical relation can be used:

$$\frac{x_f^+ - x_f}{x_D - x_C} - 2 \frac{x_f^+ - x_C}{x_D - x_C} = 2f \quad (3.85)$$

giving the final form of $\tilde{\phi}_C$:

$$\tilde{\phi}_C = 1 - \frac{(\nabla \phi)_f \cdot \hat{\mathbf{d}}}{2 (\nabla \phi)_C \cdot \hat{\mathbf{d}}} = 1 - \frac{(\nabla \phi)_f \cdot \mathbf{d}}{2 (\nabla \phi)_C \cdot \mathbf{d}} \quad (3.86)$$

There is no reference to the "far upstream" node U , or even to the previous upstream face. The product $(\nabla\phi)_f \cdot \mathbf{d}$ is calculated directly from the values of the variable on each side of the face:

$$(\nabla\phi)_f \cdot \mathbf{d} = \phi_D - \phi_C. \quad (3.87)$$

Implementation of the NVD criterion on any mesh is now simple. The information about the topological structure of the mesh is irrelevant. Even the existence of the opposite face f^- is not required – it is only necessary to have the point C in the middle of the control volume.

3.4.3 Gamma Differencing Scheme

The following Section gives the description of the Gamma differencing scheme.

CD is used in the bulk of the domain. For the values of $\tilde{\phi}_C < 0$ and $\tilde{\phi}_C > 1$ the boundedness criterion prescribes the use of UD in order to guarantee boundedness. Therefore, some sort of "switching" between the schemes is needed. This causes the most serious problems with the family of NVD differencing schemes – there is no guarantee that a steady-state solution will be obtained because of the perturbations introduced by "switching". In order to overcome this difficulty, the majority of existing NVD schemes use some form of linear upwinding in the range of values $0 < \tilde{\phi}_C < \beta_m$, where β_m is a constant of the differencing scheme, usually around 1/6 (Gaskell and Lau [49]). The meaning of β_m will become clear from the further discussion, Section 3.4.3.1. Numerical experiments show that LUD is not really appropriate for this purpose, because it causes both distortion and the shift of the profile in comparison with the exact solution.

Values of $\tilde{\phi}_C$ close to 0 ($\tilde{\phi}_C > 0$) represent the profile of the dependent variable similar to the one shown in Fig. 3.13.

If the line passing through ϕ_U , ϕ_C and ϕ_D is regarded as the exact form of the bounded local profile, it can be seen that the exact face value ϕ_f is bounded by the values obtained from UD and CD. It follows that some kind of blending between the two can be used to obtain a good estimate of the face value.

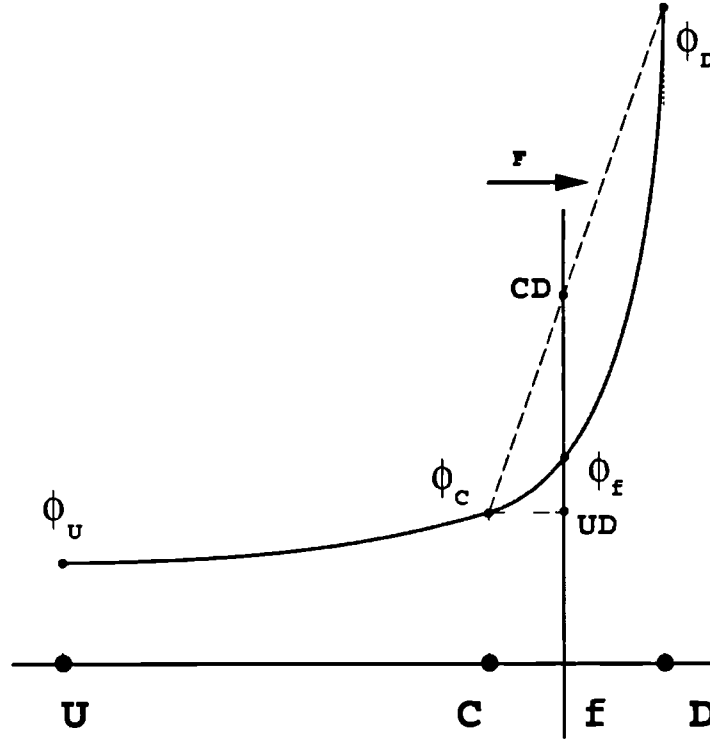


Figure 3.13: Shape of the profile for $0 < \tilde{\phi}_C < \beta_m$.

In order to establish a smooth transition, the blending between UD and CD should be used smoothly over the interval $0 < \tilde{\phi}_C < \beta_m$ and the blending factor γ should be determined on the basis of $\tilde{\phi}_C$ such that:

- $\tilde{\phi}_C = 0 \Rightarrow \gamma = 0$ (Upwind Differencing),
- $\tilde{\phi}_C = \beta_m \Rightarrow \gamma = 1$ (Central Differencing).

This practice simultaneously satisfies two important requirements:

- The face value in situations of rapidly-changing derivative of the dependent variable is well estimated and a smooth change in gradients is secured. This has proven to be much better than forms of the Linear upwinding usually used in this situation. Gaskell and Lau [49], Darwish [36].
- The transition between UD and CD is smooth. This reduces the amount of switching in the differencing scheme and improves convergence. In other NVD schemes there is a point where a small change in the value of $\tilde{\phi}_C$ causes

an abrupt change in the discretisation practice, causing the deterioration in convergence. With the Gamma differencing scheme, this is not the case.

The blending factor γ has been selected to vary linearly between $\tilde{\phi}_C = 0$ and $\tilde{\phi}_C = \beta_m$:

$$\gamma = \frac{\tilde{\phi}_C}{\beta_m}. \quad (3.88)$$

The proposed form of the Gamma differencing scheme thus includes the following.

- For all the faces of the mesh, check the direction of the flux and calculate:

$$\begin{aligned} \tilde{\phi}_C &= \frac{\phi_C - \phi_U}{\phi_D - \phi_U} \\ &= 1 - \frac{(\nabla\phi)_f \cdot \mathbf{d}}{2(\nabla\phi)_C \cdot \mathbf{d}} \end{aligned}$$

from the previous available solution or the initial guess.

- Based on $\tilde{\phi}_C$, calculate $\tilde{\phi}_f$ as:

– $\tilde{\phi}_C \leq 0$ or $\tilde{\phi}_C \geq 1$ use UD:

$$\tilde{\phi}_f = \tilde{\phi}_C \Rightarrow \phi_f = \phi_C, \quad (3.89)$$

– $\beta_m \leq \tilde{\phi}_C < 1$ use CD:

$$\tilde{\phi}_f = \frac{1}{2} + \frac{1}{2}\tilde{\phi}_C, \quad (3.90)$$

or, in terms of un-normalised variables:

$$\phi_f = f_x \phi_C + (1 - f_x) \phi_D, \quad (3.91)$$

– $0 < \tilde{\phi}_C < \beta_m$ blending is used:

$$\tilde{\phi}_f = -\frac{\tilde{\phi}_C^2}{2\beta_m} + \left(1 + \frac{1}{2\beta_m}\right) \tilde{\phi}_C, \quad (3.92)$$

or, in terms of un-normalised variables:

$$\phi_f = \left(1 - \frac{\tilde{\phi}_C}{\beta_m} (1 - f_x)\right) \phi_C + \frac{\tilde{\phi}_C}{\beta_m} (1 - f_x) \phi_D. \quad (3.93)$$

The blending factor is calculated from the local value of $\tilde{\phi}_C$ and the pre-specified constant of the scheme, β_m .

The NVD diagram for this differencing scheme is shown in Fig. 3.14.

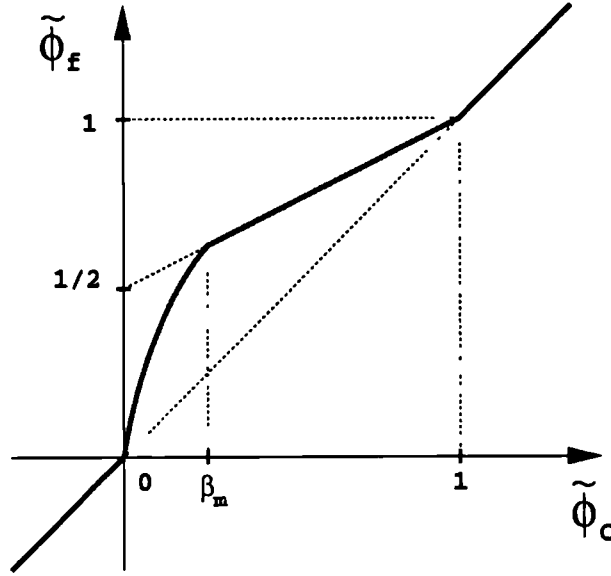


Figure 3.14: Gamma differencing scheme in the NVD diagram.

The Gamma differencing scheme uses a compact computational molecule, including one computational point on either side of each face. The calculation of $\tilde{\phi}_c$ has been modified in such a way that it does not require the far upwind node addressing. The differencing scheme can therefore be easily implemented on arbitrarily unstructured meshes. The scheme is upwind-biased not through an increase in the computational molecule, but in the way the blending factor is calculated – it can therefore be seen as a stabilised version of Central differencing.

3.4.3.1 Accuracy and Convergence of the Gamma Differencing Scheme

The role of the β_m coefficient can be seen in Fig. 3.14 – the larger the value of β_m , the more blending will be introduced. At the same time, the transition between UD and CD will be smoother. For good resolution, the value of β_m should ideally be kept as low as possible.

An upper limit on β_m comes from the accuracy requirement. For $\phi_c = \frac{1}{2}$, the variation of ϕ is linear between U and D and CD reproduces the exact value of ϕ_f . β_m should therefore always be less than $\frac{1}{2}$. Higher values of β_m result in more

numerical diffusion. For good resolution of sharp profiles, β_m should ideally be kept to about 0.1. Lower values do not improve the accuracy, but introduce the “switching” instability. The useful range of β_m is therefore:

$$\frac{1}{10} \leq \beta_m \leq \frac{1}{2}. \quad (3.94)$$

The other factor influencing the choice of β_m is the convergence for steady-state calculations. The problem of convergence with the Gamma differencing scheme is less serious than with other schemes of the switching type because of the smooth transition between UD and CD. A universal remedy for convergence problems is to increase the value of β_m (but not over $\frac{1}{2}$).

It should also be mentioned that the face value of ϕ in the Gamma differencing scheme does not in any way depend on the Courant number. Steady-state solutions will therefore be independent of the size of the time-step used to reach it.

3.5 Solution Techniques for Systems of Linear Algebraic Equations

Let us again consider the system of algebraic equations created by the discretisation, Eqn. (3.42):

$$a_P \phi_P^n + \sum_N a_N \phi_N^n = R_P.$$

This system can be solved in several different ways. Existing solution algorithms fall into two main categories: **direct** and **iterative** methods. Direct methods give the solution of the system of algebraic equations in a finite number of arithmetic operations. Iterative methods start with an initial guess and then continue to improve the current approximation of the solution until some “solution tolerance” is met. While direct methods are appropriate for small systems, the number of operations necessary to reach the solution raises with the number of equations squared, making them prohibitively expensive for large systems (Muzaferija [97]). Iterative methods are more economical, but they usually pose some requirements on the matrix.

The matrix resulting from Eqn. (3.42) is sparse – most of the matrix coefficients are equal to zero. If it were possible to choose a solver which preserves the sparsity pattern of the matrix, the computer memory requirements would be significantly decreased. Unlike direct solvers, some iterative methods preserve the sparseness of the original matrix. These properties make the use of iterative solvers very attractive.

Iterative solvers require diagonal dominance in order to guarantee convergence. A matrix is said to be diagonally equal if the magnitude of the diagonal (central) coefficient is equal to the sum of magnitudes of off-diagonal coefficients. The additional condition for diagonal dominance is that $|a_P| > \sum_n |a_N|$ for at least one row of the matrix.

In order to improve the solver convergence, it is desirable to increase the diagonal dominance of the system. Discretisation of the linear part of the source term, Eqn. (3.36), is closely related to this issue. If $S_p < 0$, its contribution increases diagonal dominance and S_p is included into the diagonal. In the case of $S_p > 0$, diagonal dominance would be decreased. It is more effective to include this term into the source and update it when the new solution is available. This measure is, however, not sufficient to guarantee the diagonal dominance of the matrix.

The analysis of the structure of the matrix brings us back to the issue of boundedness. The sufficient boundedness criterion for systems of algebraic equations mentioned in Section 3.3.1 states that the boundedness of the solution will be preserved for diagonally equal systems of equations with positive coefficients. This allows us to examine the discretised form of all the terms in the transport equation from the point of view of boundedness and diagonal dominance and identify the troublesome parts of discretisation.

The convection term creates a diagonally equal matrix only for Upwind Differencing. Any other differencing scheme will create negative coefficients, violate the diagonal equality and potentially create unbounded solution. In the case of Central Differencing on a uniform mesh, the problem is further complicated by the fact that the central coefficient is equal to zero. In order to improve the quality of the matrix for high order differencing schemes, Khosla and Rubin [73] propose

a **deferred correction implementation** for the convection term. Here, any differencing scheme is treated as an upgrade of UD. The part of the convection term corresponding to UD is treated implicitly (*i.e.* built into the matrix) and the other part is added into the source term. This, however, does not affect the boundedness in spite of the fact that the matrix is now diagonally equal, as the “troublesome” part of the discretisation still exists in the source term.

The diffusion term creates a diagonally equal matrix only if the mesh is orthogonal. On non-orthogonal meshes, the second term in Eqn. (3.34) introduces the “second neighbours” of the control volume into the computational molecule with negative coefficients, thus violating diagonal equality. As a consequence of mesh non-orthogonality, the boundedness of the solution cannot be guaranteed. The increase in the computational molecule would result in a higher number of non-zero matrix coefficients, implying a considerable increase in the computational effort required to solve the system. On the other hand, the non-orthogonal correction is usually small compared to the implicit part of the diffusion term. It is therefore reasonable to treat it through the source term. In this study, the diffusion term will therefore be split into the implicit orthogonal contribution, which includes only the first neighbours of the cell and creates a diagonally equal matrix and the non-orthogonal correction, which will be added to the source. If it is important to resolve the non-orthogonal parts of the diffusion operators (like in the case of the pressure equation, see Section 3.8), non-orthogonal correctors are included. The system of algebraic equations, Eqn. (3.42), will be solved several times. Each new solution will be used to update the non-orthogonal correction terms, until the desired tolerance is met. It should again be noted that this practice only improves the quality of the matrix but does not guarantee boundedness. If boundedness is essential, the non-orthogonal contribution should be discarded, thus creating a discretisation error described in Section 3.6.

At this point, the difference between the non-orthogonality treatments proposed in Section 3.3.1.3 becomes apparent. The decomposition of the face area vector into the orthogonal and non-orthogonal part determines the split between the implicit

and explicit part of the term, with the consequences on the accuracy and convergence of non-orthogonal correctors. The comparison of different treatments is based on several criteria:

- On which angle of non-orthogonality is it necessary to introduce non-orthogonal correctors – how good an approximation of the converged solution can be obtained after only one solution of the system?
- How many non-orthogonal correctors are needed to meet a certain tolerance?
- How does the number of solver iterations change with the number of correctors?
- If the non-orthogonal correction needs to be discarded for the sake of boundedness, which approach causes the smallest discretisation error?

The discretisation error for the diffusion term will be derived in Section 3.6. Numerical results for the convergence of three non-orthogonality treatments will be given in Section 3.7.

The discretisation of the temporal derivative creates only the diagonal coefficient and a source term contribution, thus increasing the diagonal dominance. Unfortunately, the sufficient boundedness criterion cannot be used to establish the boundedness of the discretisation, as it does not take into account the influence of the source term.

The above discussion concentrates on the analysis of the discretisation on a term-by-term basis. In reality, all of the above coefficients contribute to the matrix, thus influencing the properties of the system. It has been shown that the only terms that actually enhance the diagonal dominance are the linear part of the source and the temporal derivative.

In steady-state calculations, the beneficial influence of the temporal derivative on the diagonal dominance does not exist. In order to enable the use of iterative solvers, the diagonal dominance needs to be enhanced in some other way, namely through

under-relaxation. Consider the original system of equations, Eqn. (3.42):

$$a_P \phi_P^n + \sum_N a_N \phi_N^n = R_P.$$

Diagonal dominance is created through an artificial term added to both left and right-hand side of Eqn. (3.42):

$$a_P \phi_P^n + \frac{1-\alpha}{\alpha} a_P \phi_P^n + \sum_N a_N \phi_N^n = R_P + \frac{1-\alpha}{\alpha} a_P \phi_P^o, \quad (3.95)$$

or

$$\frac{a_P}{\alpha} \phi_P^n + \sum_N a_N \phi_N^n = R_P + \frac{1-\alpha}{\alpha} a_P \phi_P^o. \quad (3.96)$$

Here, ϕ_P^o here represents the value of ϕ from the previous iteration and α is the under-relaxation factor ($0 < \alpha \leq 1$). Additional terms cancel out when steady-state is reached ($\phi_P^n = \phi_P^o$).

In this study, the iterative solution procedure used to solve the system of algebraic equations is the Conjugate Gradient (CG) method, originally proposed by Hestens and Steifel [63]. It guarantees that the exact solution will be obtained in the number of iterations smaller or equal to the number of equations in the system. The convergence rate of the solver depends on the dispersion of the eigenvalues of the matrix $[A]$ in Eqn. (3.43) and can be improved through pre-conditioning. For symmetric matrices, the Incomplete Cholesky preconditioned Conjugate Gradient (ICCG) solver will be used. The method is described in detail by Jacobs, [67]. The adopted solver for asymmetric matrices is the Bi-CGSTAB by van der Vorst [136].

3.6 Numerical Errors in the Discretisation Procedure

This Section gives an overview of numerical errors introduced through the discretisation procedure. Errors come from two sources. The first group of errors is a consequence of discretisation practices that are lower than second-order accurate. This includes the discretisation of temporal terms and convection differencing schemes,

where it was necessary to lower the order of discretisation in order to preserve the boundedness of the solution. The second group of errors is caused by the discretisation of the solution domain. They are a direct consequence of mesh quality. This group of errors includes insufficient mesh resolution, skewness and non-orthogonality errors.

The Finite Volume method described above is characterised as second-order accurate and therefore also creates higher-order errors. Since the method has been developed for second-order continuum mechanics problems, the higher-order errors will be neglected⁴.

3.6.1 Numerical Diffusion from Convection Differencing Schemes

The second-order numerical error in the Finite Volume method can be described numerical diffusion. It is a consequence of the discretisation practice which is not fully second-order accurate, selected to preserve the boundedness of the solution. The change in accuracy of the convection discretisation can be examined by comparing the selected differencing scheme, with the second-order (unbounded) Central Differencing (CD). Bounded differencing schemes produce a numerical diffusion-type term of the form:

$$\nabla \cdot (\Gamma_N \cdot \nabla \phi), \quad (3.97)$$

where Γ_N is the numerical diffusion tensor.

Every bounded differencing schemes of the TVD and NVD family can be written as a sum of Central Differencing and the numerical diffusion term of the above form, possibly with some higher-order terms in the case when the original (unbounded) differencing scheme is more than second-order accurate. Here, we shall present a method for the evaluation of the numerical diffusion for Blended Differencing, Eqn. 3.23, and examine the properties of the numerical diffusion term

⁴An extension of the FVM to fourth-order accuracy can be found in Lilek and Perić [88]. An illustration of the behaviour of higher-order errors has been presented by Leonard [83].

Consider the face value of ϕ obtained by CD, Eqn. (3.19) and BD, Eqn. (3.23):

$$\begin{aligned}(\phi_f)_{CD} &= f_x \phi_P + (1 - f_x) \phi_N, \\(\phi_f)_{BD} &= [(1 - \gamma) \max(\text{sgn}(F), 0) + \gamma f_x] \phi_P \\&\quad + [(1 - \gamma) \min(\text{sgn}(F), 0) + \gamma(1 - f_x)] \phi_N.\end{aligned}$$

The difference in the face value is:

$$\begin{aligned}\delta\phi_f - (\phi_f)_{CD} - (\phi_f)_{BD} \\&= [(1 - \gamma) f_x - (1 - \gamma) \max(\text{sgn}(F), 0)] \phi_P \\&\quad + [(1 - \gamma) (1 - f_x) - (1 - \gamma) \min(\text{sgn}(F), 0)] \phi_N.\end{aligned}\tag{3.98}$$

To clarify the meaning of this expression, let us assume that $F > 0$:

$$\begin{aligned}\delta\phi_f &= [(1 - \gamma) f_x - (1 - \gamma)] \phi_P + (1 - \gamma) (1 - f_x) \phi_N \\&= (1 - \gamma) (1 - f_x) (\phi_N - \phi_P).\end{aligned}\tag{3.99}$$

For the whole convection term, the difference can be written as:

$$\begin{aligned}E_c &= \sum_f F \delta\phi_f = \sum_f \mathbf{S} \cdot (\rho \mathbf{U})_f [(1 - \gamma) (1 - f_x) (\phi_N - \phi_P)] \\&= \sum_f \mathbf{S} \cdot [(1 - \gamma) (1 - f_x) (\rho \mathbf{U})_f \mathbf{d} \cdot (\nabla \phi)_f] \\&= \sum_f \mathbf{S} \cdot [((1 - \gamma) (1 - f_x) (\rho \mathbf{U})_f \mathbf{d}) \cdot (\nabla \phi)_f] \\&= \sum_f \mathbf{S} \cdot (\Gamma_N \cdot \nabla \phi)_f,\end{aligned}\tag{3.100}$$

where Γ_N is a tensorial numerical viscosity:

$$\Gamma_N = (1 - \gamma) (1 - f_x) (\rho \mathbf{U})_f \mathbf{d}.\tag{3.101}$$

More generally, the numerical diffusion tensor can be written as:

$$\Gamma_N = (1 - \gamma) \rho \mathbf{U}_f \mathbf{d} [(1 - f_x) \max(\text{sgn}(F), 0) + f_x \min(\text{sgn}(F), 0)].\tag{3.102}$$

Eqs. (3.100) and (3.102) give us an insight into the properties of numerical diffusion. Firstly, it can be seen that the term is indeed diffusion-like, depending on

gradients of ϕ . For a uniform distribution of ϕ , this term vanishes. Other important facts are that the term scales linearly with the mesh-spacing (*i.e.* with $|\mathbf{d}|$), that it is large in the cases of large convection velocity and that it disappears when $(\mathbf{U})_f \cdot \mathbf{S}$ goes to zero (this is usually called “mesh-to-flow alignment”). For Central Differencing ($\gamma = 1$), Γ_N also goes to zero. The numerical diffusion tensor for Upwind Differencing on a uniform mesh ($f_x = \frac{1}{2}$) can be simplified to:

$$\Gamma_N = \frac{1}{2}(\rho \mathbf{U})_f \mathbf{d}. \quad (3.103)$$

Numerical diffusion for the Gamma differencing scheme (Section 3.4) can also be calculated from Eqn. (3.102). The blending factor is calculated from the face value of $\tilde{\phi}_C$ as:

$$\gamma = \begin{cases} \gamma = 0 & \text{for } \tilde{\phi}_C \leq 0 \text{ and } \tilde{\phi}_C \geq 1, \\ \gamma = \frac{\tilde{\phi}_C}{\beta_m} & \text{for } 0 < \tilde{\phi}_C < \beta_m, \\ \gamma = 1 & \text{for } \beta_m \leq \tilde{\phi}_C < 1. \end{cases} \quad (3.104)$$

The error coming from numerical diffusion has been well-known. Many attempts to reduce its influence on the solution have been made, mainly by reducing the effective blending factor. The effects of numerical diffusion on the accuracy of the solution will be illustrated in Section 3.7.

3.6.2 Numerical Diffusion from Temporal Discretisation

A detailed analysis of the temporal discretisation shows that it can also produce a numerical diffusion error. This is important for transient calculations – the development of the solution in time can be severely smeared. The Crank-Nicholson method, Eqn. 3.41, gives a fully second-order accurate discretisation in time. In order to derive the numerical diffusion term for any other temporal discretisation scheme, we need only to compare it with the Crank-Nicholson method. Let us first consider the **Euler Implicit temporal discretisation**.

The comparison between the Crank-Nicholson discretisation, Eqn. (3.41):

$$\begin{aligned} \frac{\rho_P \phi_P^n - \rho_P \phi_P^o}{\Delta t} V_P + \frac{1}{2} \sum_f F \phi_f^n - \frac{1}{2} \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f^n \\ + \frac{1}{2} \sum_f F \phi_f^o - \frac{1}{2} \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f^o \\ = SuV_P + \frac{1}{2} SpV_P \phi_P^n + \frac{1}{2} SpV_P \phi_P^o \end{aligned}$$

and the Euler Implicit method, Eqn. (3.50):

$$\begin{aligned} \frac{\rho_P \phi_P^n - \rho_P \phi_P^o}{\Delta t} V_P + \sum_f F \phi_f^n - \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f^n \\ = SuV_P + SpV_P \phi_P^n, \end{aligned}$$

gives the difference of

$$\begin{aligned} E_t = -\frac{1}{2} \left[\sum_f F \phi_f^n - \sum_f F \phi_f^o \right] \\ + \frac{1}{2} \left[\sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f^n - \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f^o \right] \\ - \frac{1}{2} (SpV_P \phi_P^n - SpV_P \phi_P^o). \end{aligned} \quad (3.105)$$

Using Eqn. (3.4):

$$\phi^n = \phi^o + \frac{\partial \phi}{\partial t} \Delta t$$

and

$$(\nabla \phi)^n = (\nabla \phi)^o + \frac{\partial (\nabla \phi)}{\partial t} \Delta t, \quad (3.106)$$

Eqn. (3.105) can be written as:

$$\begin{aligned} E_t = -\frac{\Delta t}{2} \sum_f F \left(\frac{\partial \phi}{\partial t} \right)_f \\ + \frac{\Delta t}{2} \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot \frac{\partial (\nabla \phi)_f}{\partial t} \\ - \frac{\Delta t}{2} SpV_P \left(\frac{\partial \phi}{\partial t} \right)_P. \end{aligned} \quad (3.107)$$

E_t can be decomposed into three parts:

- Convection error:

$$D_C = -\frac{\Delta t}{2} \sum_f F \left(\frac{\partial \phi}{\partial t} \right)_f, \quad (3.108)$$

- Diffusion error:

$$D_D = \frac{\Delta t}{2} \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot \frac{\partial \nabla \phi}{\partial t}_f, \quad (3.109)$$

- Source term error:

$$D_S = -\frac{\Delta t}{2} S_P V_P \left(\frac{\partial \phi}{\partial t} \right)_P. \quad (3.110)$$

In order to perform the analysis of E_t , it is necessary to express the temporal derivative of ϕ as a combination of spatial terms. The original transport equation will be used in this form:

$$\frac{\partial(\rho\phi)}{\partial t} = S_\phi(\phi) - \nabla \cdot (\rho \mathbf{U} \phi) + \nabla \cdot (\rho \Gamma_\phi \nabla \phi). \quad (3.111)$$

The convection term can be split into two parts:

$$\nabla \cdot (\rho \mathbf{U} \phi) = \rho \mathbf{U} \cdot \nabla \phi + \phi \nabla \cdot (\rho \mathbf{U}). \quad (3.112)$$

In order to simplify the derivation, it is assumed that $\rho = \text{const}$. It follows:

$$\frac{\partial \phi}{\partial t} = \frac{S_\phi(\phi)}{\rho} - \mathbf{U} \cdot \nabla \phi - \phi \nabla \cdot \mathbf{U} + \nabla \cdot (\Gamma_\phi \nabla \phi). \quad (3.113)$$

Using Eqn. (3.113), the convection error, Eqn. (3.108), splits into four terms:

$$D_{C1} = -\frac{\Delta t}{2} \sum_f F \left[\frac{S_\phi(\phi)}{\rho} \right]_f, \quad (3.114)$$

$$D_{C2} = \frac{\Delta t}{2} \sum_f F [\mathbf{U} \cdot \nabla \phi]_f, \quad (3.115)$$

$$D_{C3} = \frac{\Delta t}{2} \sum_f F [\phi \nabla \cdot \mathbf{U}]_f, \quad (3.116)$$

$$D_{C4} = \frac{\Delta t}{2} \sum_f F [\nabla \cdot \Gamma_\phi \nabla \phi]_f. \quad (3.117)$$

D_{C1} cannot be further analysed, as it depends on the distribution of the source. It will be shown that D_{C2} has an important effect it can be transformed into

a numerical diffusion-type term. D_{C3} scales with the divergence of velocity. In standard situations $\nabla \cdot \mathbf{U}$ is considered to be small and this term is neglected. D_{C4} includes higher derivatives of ϕ and is therefore neglected.

Let us now come back to D_{C2} , Eqn. (3.115). Following a procedure similar to the one in Section 3.6.1, this term can be written as:

$$D_{C2} = \frac{\Delta t}{2} \sum_f \mathbf{S} \cdot [(\rho \mathbf{U} \mathbf{U}) \cdot \nabla \phi]_f. \quad (3.118)$$

A closer look into Eqn. (3.118), reveals that this term has the form $\nabla \cdot (\Gamma_T \cdot \nabla \phi)$ where Γ_T is a form of tensorial diffusion. This (numerical) diffusion has a maximum if vectors $\mathbf{S}$, $\mathbf{U}$ and $\nabla \phi$ are parallel:

$$D_{C2} = \sum_f \mathbf{S} \cdot \left[\frac{\Delta t}{2} \rho |\mathbf{U}|^2 (\nabla \phi)_f \right]. \quad (3.119)$$

This would correspond to a one-dimensional transport of a wave. The maximum equivalent numerical diffusion is therefore:

$$(\Gamma_T)_{max} = \frac{\Delta t}{2} \rho |\mathbf{U}|^2. \quad (3.120)$$

Eqn. (3.120) can be written in terms of the Courant number, Eqn. (3.48):

$$(\Gamma_T)_{max} = \frac{1}{2} Co \rho |\mathbf{U}| \mathbf{d}. \quad (3.121)$$

Comparison of Eqs. (3.121) and (3.103) shows that for $Co = 1$ the numerical diffusion from the temporal discretisation is the same as numerical diffusion from UD. This effect is far from negligible. For good temporal accuracy, a higher-order temporal scheme is as important as a higher order convection differencing scheme.

Following the equivalent path for the diffusion error D_D , Eqn. (3.109), it can be shown that it produces terms that either cannot be analysed or depend on higher gradients of ϕ . For the similar reason, the source error D_S cannot be analysed either.

An equivalent analysis for the **explicit temporal discretisation** produces a similar result. The main error term comes from the convection part of the decomposition. It can be written as the diffusion term equivalent to Eqn. (3.119), but with numerical diffusion of the opposite sign:

$$(\Gamma_T)_{max} = -\frac{1}{2} Co \rho |\mathbf{U}| \mathbf{d}. \quad (3.122)$$

Eqs. (3.102) and (3.122) can also be used to confirm some well-known stability analysis results. It is, for example, well known that explicit Central Differencing is an unconditionally unstable combination of spatial and temporal discretisation for transport equations without a diffusion term. According to Eqn. (3.103), CD does not introduce any numerical diffusion in the system. The numerical diffusion from the Explicit discretisation in time is negative, producing negative effective diffusion. Any system with negative diffusion is unstable. In the case of explicit Upwind Differencing, the effective diffusion in the system is positive if the numerical diffusion from UD is larger than the negative diffusion from the temporal discretisation. This is the case if $Co < 1$, which agrees with the classical stability result (see *eg.* Hirsch [65]). For $Co > 1$, the effective diffusion becomes negative and the system becomes unstable.

Backward Differencing in time is a second-order accurate scheme. It can be shown that the additional part of the temporal derivative actually cancels out the numerical diffusion term resulting from D_C (see Appendix A). The leading term of the truncation error is, however, four times larger than for the Crank-Nicholson method. The major deficiency of the scheme comes from its extrapolative behaviour in time, which causes unboundedness. An example of its behaviour, given in Section 3.7 shows that a combination of a bounded convection differencing scheme and the Crank-Nicholson temporal discretisation creates a bounded solution, which is not the case if the Backward differencing in time is used.

3.6.3 Mesh-Induced Errors

Let us start the overview of mesh-induced errors with the **insufficient mesh resolution**. If the solution changes rapidly in some regions of the domain and the local number of computational points is not sufficient to describe that change, the shape of the solution will effectively be lost. Even if the exact solution is available in a certain number of points, the picture of the overall solution depends on the distribution and the number of those points. In Finite Volume calculations, the accuracy of interactions described by the differential equation depends on the mesh resolution.

It is therefore important that the selected computational points can provide a good picture of the solution.

For the second-order spatial discretisation, the mesh resolution error is a consequence of the assumed linear variation of the solution over the control volume. Although it does not change the nature of the differential equation that is being solved, insufficient mesh resolution may produce a completely incorrect picture of the solution.

The influence of mesh non-orthogonality on boundedness has been described in Section 3.5. The advantage of orthogonal meshes in the discretisation of the diffusion term comes from the fact that only the first neighbours of the cell are included into the computational molecule. The non-orthogonal term, irrespective of its implicit or explicit treatment, also uses the second neighbours of the cell, thus increasing the size of the computational molecule. Although both methods of discretisation are second order accurate, the increase in the computational module implies a larger leading term in the Taylor series expansion and higher errors.

Non-orthogonality error, which violates the order of discretisation comes into effect only if it is necessary to guarantee the boundedness of the solution. In this case, the non-orthogonal component in Eqn. (3.34) is discarded, thus creating a discretisation practice that obeys the sufficient boundedness criterion. For the complete diffusion term, the non-orthogonality error has the following form:

$$\begin{aligned} E_d &= \sum_f \mathbf{S}_f \cdot [(\rho \mathbf{U})_f \mathbf{k} \cdot (\nabla \phi)_f] \\ &= \nabla \cdot (\Gamma_D \nabla \phi), \end{aligned} \quad (3.123)$$

where

$$\Gamma_D = (\rho \mathbf{U})_f \mathbf{k}. \quad (3.124)$$

The magnitude of Γ_D depends both on the non-orthogonality angle of the mesh and the selected non-orthogonality approach (see Section 3.3.1.3). For the same angle of non-orthogonality, E_d will be smallest for the minimum correction approach, as

and

$$\mathbf{m} = \mathbf{x}_f - \mathbf{x}_{fi}. \quad (3.128)$$

E_s can be transformed into:

$$\begin{aligned} E_s &= \sum_f \mathbf{S} \cdot [(\rho \mathbf{U})_f \mathbf{m} \cdot (\nabla \phi)_f] \\ &= \nabla \cdot (\Gamma_S \nabla \phi), \end{aligned} \quad (3.129)$$

where

$$\Gamma_S = (\rho \mathbf{U})_f \mathbf{m}. \quad (3.130)$$

When examining the importance of Eqn. (3.129) one should keep in mind that $\mathbf{S}$ and $\mathbf{m}$ are orthogonal to each other, Fig. 3.15. On meshes of reasonable quality $|\mathbf{m}|$ is smaller than d . The influence of Eqn. (3.129) is expected to be smaller than the numerical diffusion from the convection differencing scheme. On highly distorted meshes the influence of this term can be significant.

As in the case of mesh non-orthogonality, the error in Eqn. (3.129) can be removed using the face interpolate of $\nabla \phi$. This, however, increases the size of the computational molecule and potentially causes unboundedness.

3.7 Numerical Examples

In this Section, numerical data for some of the discretisation errors will be presented. We shall start with the properties of basic convection differencing schemes. A comparison of the Gamma differencing scheme with other high-resolution differencing schemes will be given in Section 3.7.2. The behaviour of different temporal discretisation schemes is presented in Section 3.7.3. Finally, in Section 3.7.4, three non-orthogonality treatments (see Section 3.3.1.3) are tested.

3.7.1 Numerical Diffusion from Convection Discretisation

In order to present the behaviour of UD, CD and Gamma differencing scheme, the following test case has been selected. Consider a steady-state convection of a scalar

ϕ , Eqn. (3.131), in a square domain shown in Fig. 3.16:

$$\nabla \cdot (\rho \mathbf{U} \phi) = 0. \quad (3.131)$$

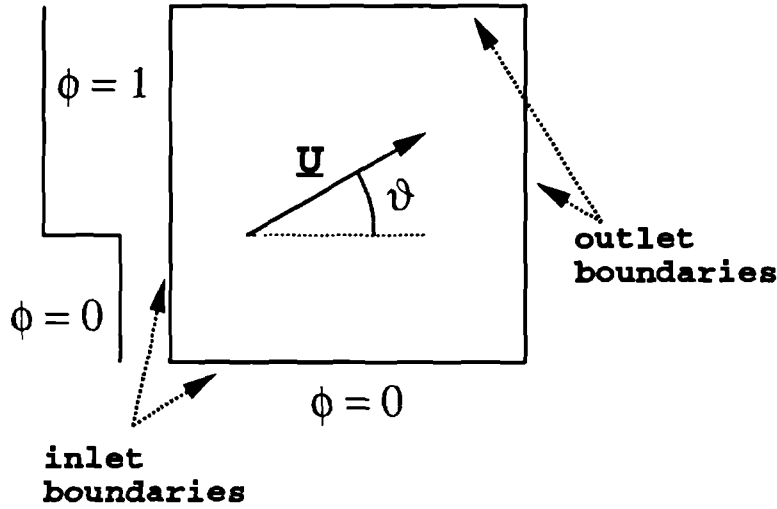


Figure 3.16: Step-profile test setup.

Three mesh-to-flow angles are examined: $\theta = 0^\circ$, $\theta = 30^\circ$ and $\theta = 45^\circ$. At the inlet boundary, a step-profile of ϕ is prescribed.

The velocity field is prescribed to be uniform. For $\theta = 0^\circ$, all three differencing schemes reproduce the exact solution, Fig. 3.17 to within one cell width.

Fig. 3.18 shows the diffusion of the profile along the domain for UD. The flow is no longer aligned with the mesh and numerical diffusion smears out the solution. The unbounded solution for CD on the same case is presented in Fig. 3.19.

In order to simplify the comparison, profiles of ϕ along the sampling line normal to the flow are plotted. Fig. 3.20 gives the comparison of UD, CD and Gamma differencing scheme. It can be seen that Gamma differencing produces a bounded and accurate solution.

Fig. 3.21 presents the same data for $\theta = 45^\circ$. CD still gives an unbounded solution, although the oscillations are now much smaller. The amount of numerical diffusion in UD is now even larger than in the worst case in terms of mesh-to-flow alignment. Following the accuracy of CD and giving a bounded solution, the

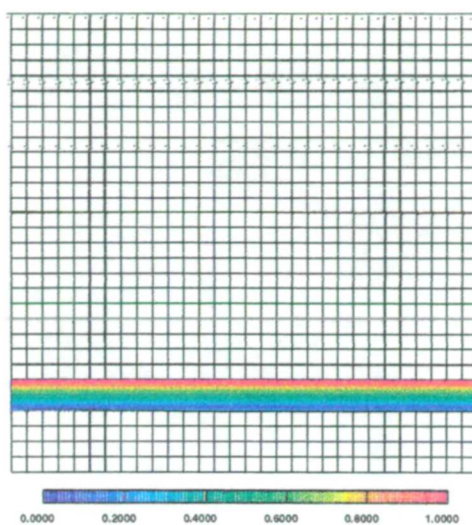


Figure 3.17: Convection of a step-profile, $\theta = 0^\circ$, UD.

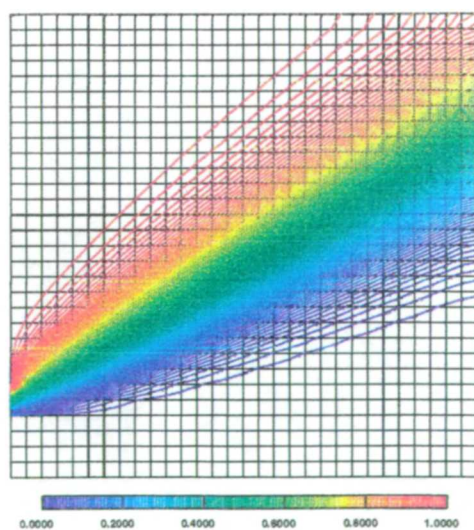


Figure 3.18: Convection of a step-profile, $\theta = 30^\circ$, UD.

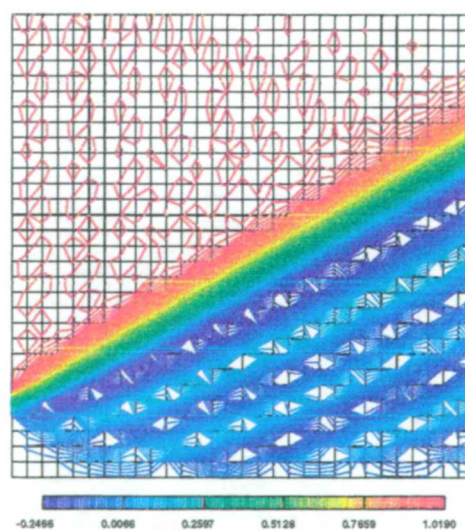


Figure 3.19: Convection of a step-profile, $\theta = 30^\circ$, CD.

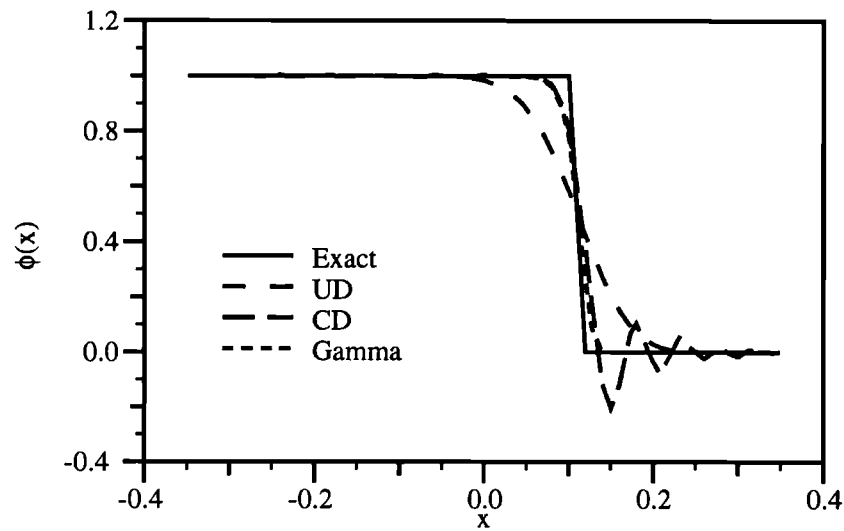


Figure 3.20: Convection of a step-profile, $\theta = 30^\circ$, UD, CD and Gamma differencing schemes.

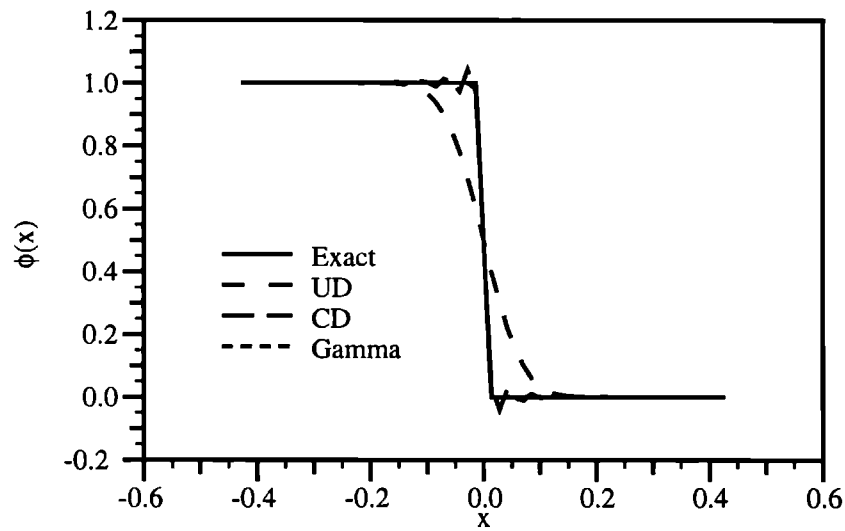


Figure 3.21: Convection of a step-profile, $\theta = 45^\circ$, UD, CD and Gamma differencing schemes.

Gamma differencing scheme reproduces the exact profile.

3.7.2 Comparison of the Gamma Differencing Scheme with Other High-Resolution Schemes

Let us now present a comparison of the Gamma differencing scheme with a group of other high-resolution schemes. The test setup is similar to Fig. 3.16, with different profiles of ϕ at the inlet boundary. The mesh to-flow angle is fixed to 30° . Following Leonard [83], three boundary profiles of ϕ are selected:

- Step-profile:

$$\phi(x) = \begin{cases} 0 & \text{for } 0 \leq x < \frac{1}{6}, \\ 1 & \text{for } \frac{1}{6} \leq x \leq 1, \end{cases} \quad (3.132)$$

- $\sin^2$ -profile:

$$\phi(x) = \begin{cases} \sin^2 \left[3\pi \left(x - \frac{1}{6} \right) \right] & \text{for } \frac{1}{6} \leq x < \frac{1}{2}, \\ 0 & \text{elsewhere,} \end{cases} \quad (3.133)$$

- Semi-ellipse:

$$\phi(x) = \begin{cases} \sqrt{1 - \left(\frac{x - \frac{1}{2}}{\frac{1}{6}} \right)^2} & \text{for } \frac{1}{6} \leq x < \frac{1}{2}, \\ 0 & \text{elsewhere.} \end{cases} \quad (3.134)$$

The comparison of the profiles is given along the line in the centre of the domain, normal to the flow. The differencing schemes included in this comparison are:

- Gamma differencing scheme, Section 3.4,
- Self-filtered Central Differencing (SFCD) [128],
- van Leer TVD scheme [139],
- SUPERBEE TVD differencing scheme by Roe [118],
- SOUCUP NVD differencing scheme by Zhu and Rodi [155],
- SMART NVD scheme by Gaskell and Lau [49].

3.7.2.1 Step-profile

The results for the first test case are shown in Figs. 3.22, 3.23 and 3.24. Several interesting features can be recognised. The step-resolution for Gamma is far superior to the older version of stabilised Central Differencing (SFCD). The Gamma differencing scheme gives the resolution of CD over the step and completely removes unphysical oscillations.

SUPERBEE, Fig. 3.23, actually gives better step-resolution than Gamma. This is the most compressive TVD differencing scheme. While SUPERBEE is very good for sharp profiles, distortions of smooth solutions are too large to be accepted (Leonard [83]).

SOUCUP, Fig. 3.24, is more diffusive than Gamma. The other NVD scheme, SMART, gives a solution very close to the one obtained from the Gamma differencing scheme. One should have in mind that SMART is actually a stabilised version of QUICK by Leonard, [81]. QUICK uses a computational molecule which is twice the size of the one used by CD and Gamma. Apart from that, SMART is usually implemented in a deferred correction mode, causing distortions in the temporal behaviour. The Gamma differencing scheme, with its compact computational molecule, does not require such treatment.

3.7.2.2 $\sin^2$ -profile

The results for the convection of a $\sin^2$ profile are shown in Figs. 3.25, 3.26 and 3.27. This test case has been selected to present the behaviour of the differencing schemes on smoothly changing profiles. Another interesting feature of this test is a one-point peak in the exact solution, Fig. 3.25. It is known that TVD schemes reduce to first-order accuracy in the vicinity of extrema. This phenomenon is usually called “clipping”. The amount of “clipping” gives us a direct indication of the quality of the differencing scheme.

SFCD, Fig. 3.25, shows a considerable amount of “clipping”. The maximum of the profile for the Gamma differencing scheme is about 0.85 – considerably less than

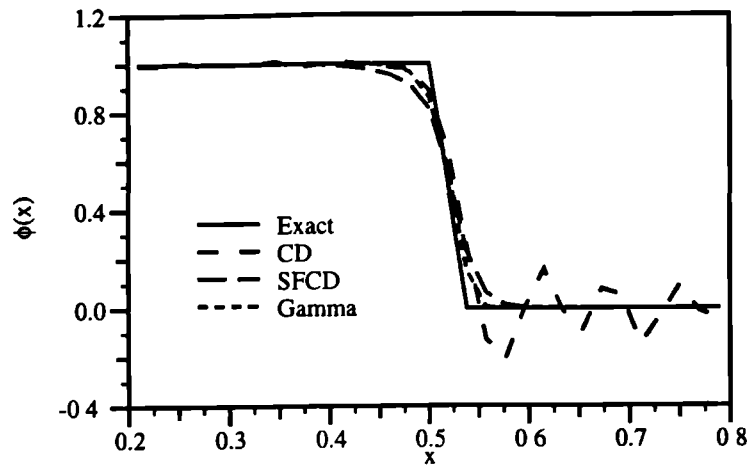


Figure 3.22: Convection of a step-profile, $\theta = 30^\circ$, CD, SFCD and Gamma differencing schemes.

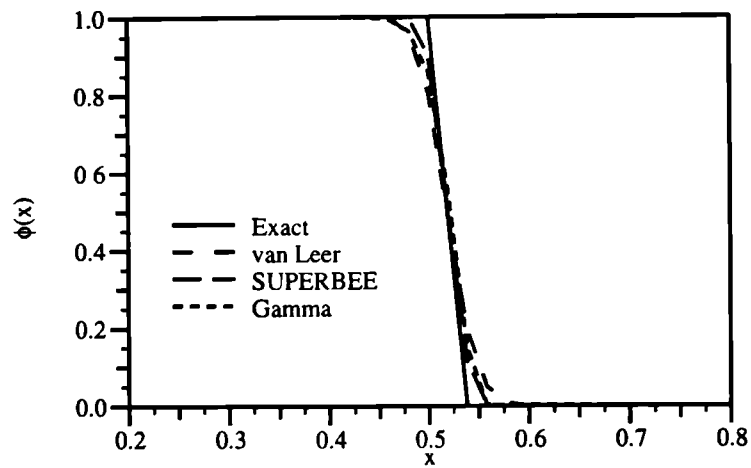


Figure 3.23: Convection of a step-profile, $\theta = 30^\circ$, van Leer, SUPERBEE and Gamma differencing schemes.

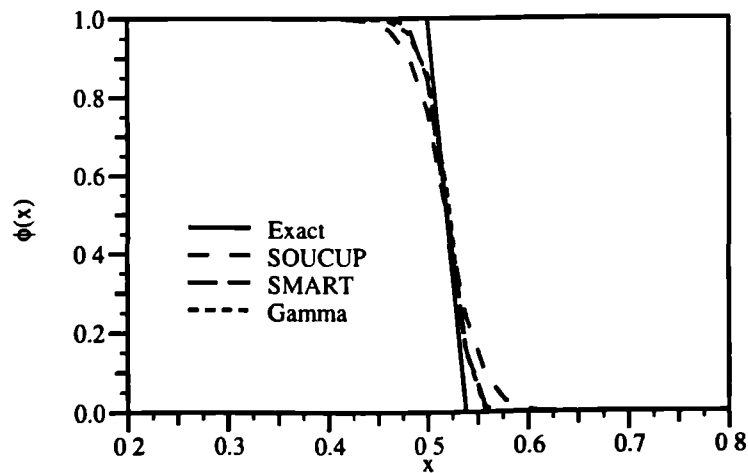


Figure 3.24: Convection of a step-profile, $\theta = 30$, SOUCUP, SMART and Gamma differencing schemes.

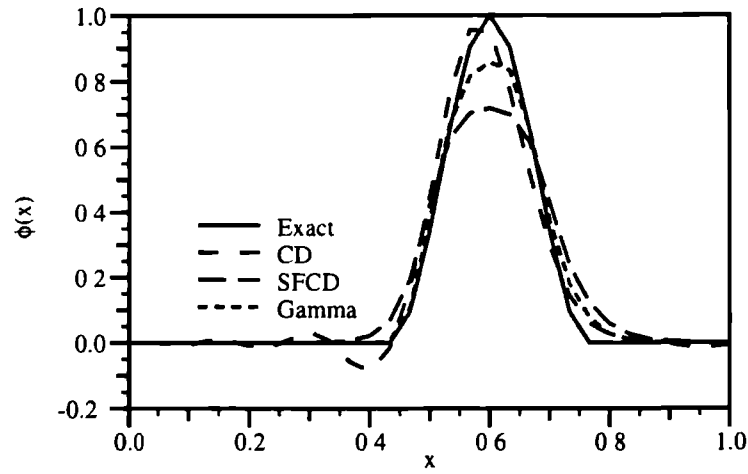


Figure 3.25: Convection of a $\sin^2$ -profile, $\theta = 30$, CD, SFCD and Gamma differencing schemes.

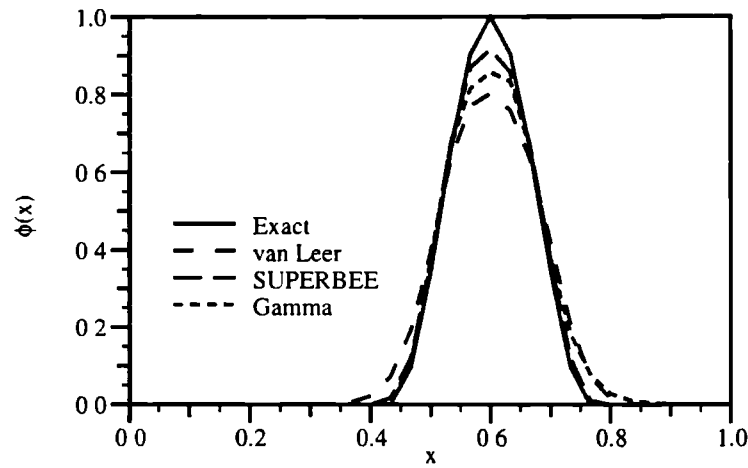


Figure 3.26: Convection of a $\sin^2$ -profile, $\theta = 30$, van Leer, SUPERBEE and Gamma differencing schemes.

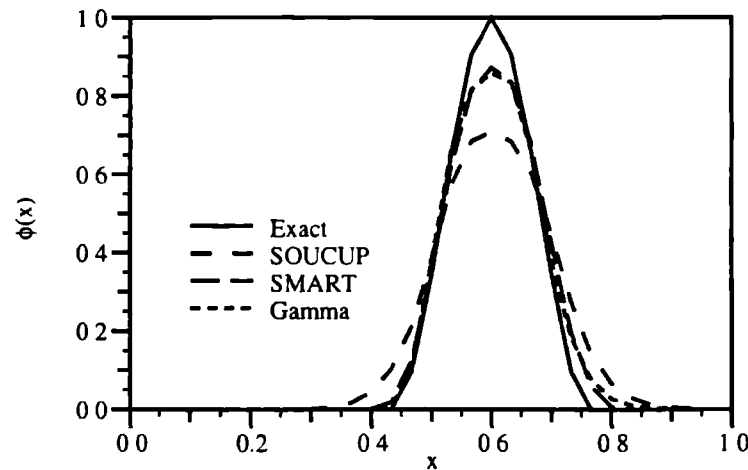


Figure 3.27: Convection of a $\sin^2$ -profile, $\theta = 30$, SOUCUP, SMART and Gamma differencing schemes.

that for CD. This is the price that needs to be paid for a bounded solution.

The best of the TVD schemes is again SUPERBEE, Fig. 3.26. It follows the exact profile quite accurately and peaks at about $\phi = 0.9$. The van Leer TVD limiter still seems to be too diffusive.

Fig. 3.27 shows the comparison for the NVD schemes. Again, there is not much difference between SMART and Gamma solutions. SOUCUP still seems to be too diffusive.

3.7.2.3 Semi-ellipse

The results for the convection of a semi-elliptic profile with the selected differencing schemes are shown in Figs. 3.28, 3.29 and 3.30. The main feature of this test-case is an abrupt change in the gradient on the edges of the semi-ellipse, followed by the smooth change in ϕ . CD, Fig. 3.28, introduces oscillations in the solution, influencing the smooth region of the profile. This is usually called “waviness” (Leonard [83]). Gamma again proves to be a more effective way of stabilising CD than SFCD.

Fig. 3.29 clearly shows the problems with SUPERBEE – it tries to transform the semi-ellipse into a top-hat profile, as has been noted by Leonard [83]. Van Leer’s limiter performs quite well in this situation. This can be attributed to the balanced introduction of numerical diffusion and anti-diffusion used in this limiter.

The solutions for SMART and Gamma, Fig. 3.30 are again quite close: SMART gives marginally smaller “clipping” of the top and Gamma follows the smooth part of the profile better. SOUCUP, while describing the smooth profile well, clips off about 10% of the peak.

3.7.3 Numerical Diffusion from Temporal Discretisation

The objective of the following examples is to show the influence of different forms of temporal discretisation on the accuracy of the solution. Three test cases have been selected:

- 1-D transport of a step profile,

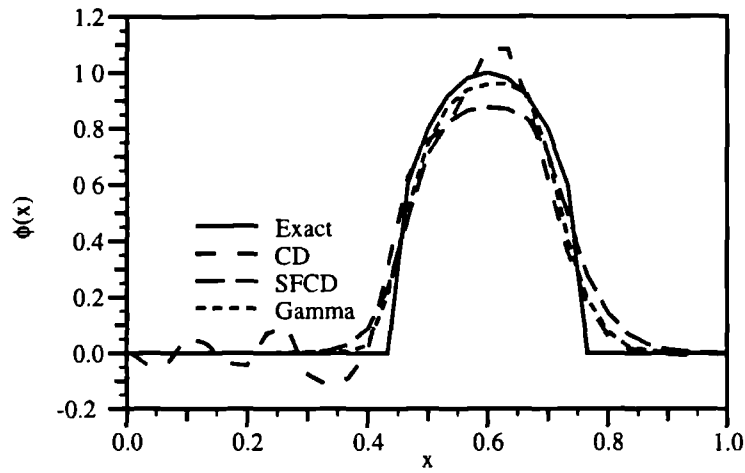


Figure 3.28: Convection of a semi-ellipse, $\theta = 30^\circ$, CD, SFCD and Gamma differencing schemes.

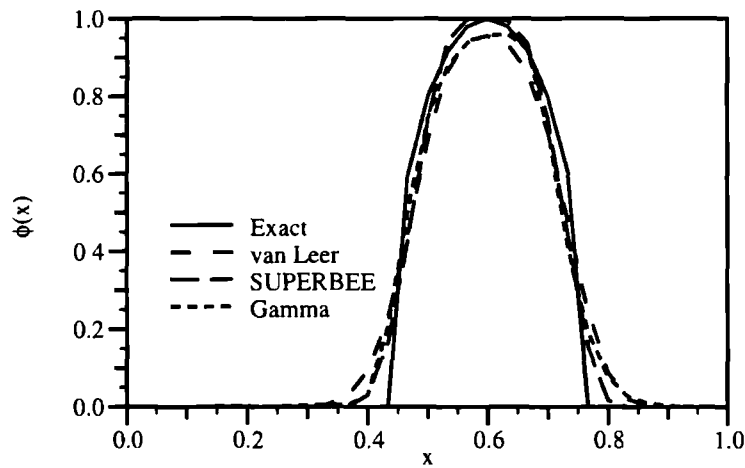


Figure 3.29: Convection of a semi-ellipse, $\theta = 30^\circ$, van Leer, SUPERBEE and Gamma differencing schemes.

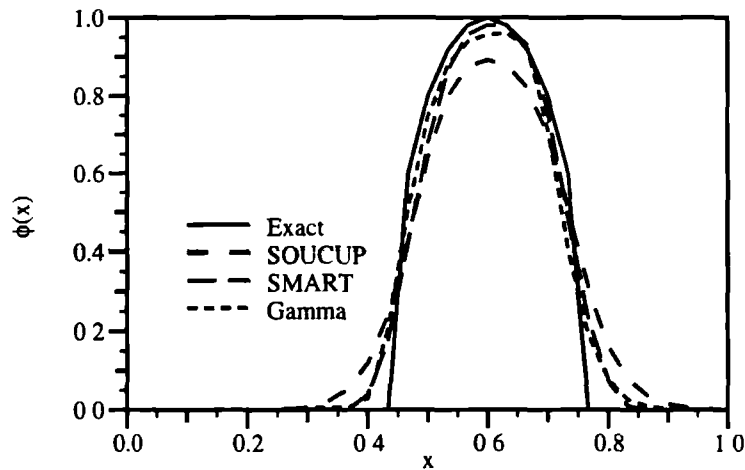


Figure 3.30: Convection of a semi-ellipse, $\theta = 30^\circ$, SOUCUP, SMART and Gamma differencing schemes.

- 1-D transport of a half $s n^2$ profile,
- 2-D transport of a “bubble”.

In order to minimise the effects of convection discretisation, the Gamma differencing scheme has been used for both 1-D test cases. It is known that the Crank-Nicholson temporal discretisation is second-order accurate. This solution can therefore be used to illustrate how much of the error comes from the convection differencing scheme. It is, however, still easy to recognise the features of the temporal discretisation error.

3.7.3.1 1-D Tests

The test setup consists of a one-dimensional domain with the constant velocity field. The initial profile is prescribed at the inlet and convected down the domain. For all test cases, the Courant number is set to $Co = 0.2$.

Figs. 3.31 and 3.32 present solutions for four different methods of temporal discretisation:

- Euler Implicit discretisation, Eqn. (3.44),
- Explicit discretisation, Eqn. (3.47),
- Crank-Nicholson method, Eqn. (3.41),
- Backward Differencing in time, Eqn. (3.55).

In Section 3.6.2 the numerical diffusion coefficients for the Euler Implicit and Explicit temporal discretisation have been derived. Fig. 3.31 illustrates this behaviour. The Euler Implicit discretisation introduces positive numerical diffusion, resulting in the smeared profile. The Explicit discretisation actually gives better step resolution as a consequence of negative numerical diffusion from temporal discretisation. Crank-Nicholson and Backward Differencing in time give essentially the same result. Inaccuracy of Backward Differencing is larger because of its extrapolative behaviour the leading term of Taylor series truncation error scales with $4\Delta t^2$, compared with

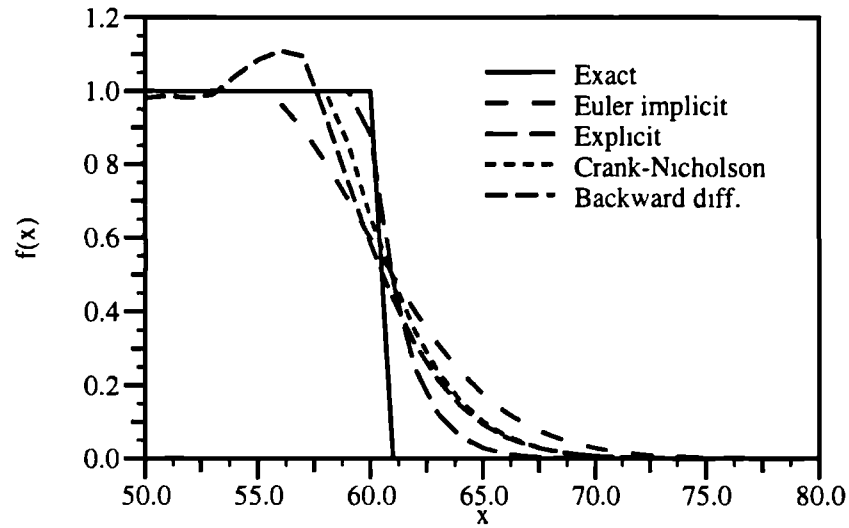


Figure 3.31: Transport of a step-profile after 300 time-steps, four methods of temporal discretisation.

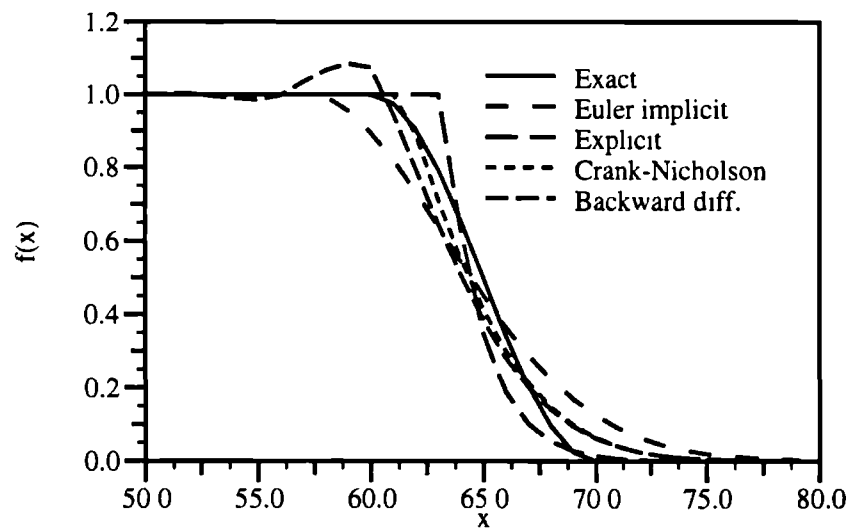


Figure 3.32: Transport of a half- s^2 profile after 300 time-steps, four methods of temporal discretisation.

Δt^2 for Crank-Nicholson. The Backward Differencing solution is also unbounded in spite of the fact that a bounded convection differencing scheme has been used.

The effects of negative numerical diffusion for Explicit temporal discretisation can be clearly seen in Fig. 3.32. Comparison with the exact solution shows that the distribution of ϕ is steeper than it should be. Both Crank-Nicholson and Backward Differencing in time follow the smooth profile well.

3.7.3.2 2-D Transport of a “Bubble”

In order to show the effects of the tensorial numerical viscosity from temporal discretisation in space, a two-dimensional test-case has been selected. Consider a transport of a “bubble” of a passive scalar in a uniform velocity field, Fig. 3.33. In order to remove the effects of convection discretisation, a special “interface-tracking” differencing scheme has been used. It will guarantee that both boundedness and the shape of the “bubble” are not affected. The InterGamma differencing scheme is particularly suitable for surface tracking. Details on this highly compressive and bounded differencing scheme can be found in [69].

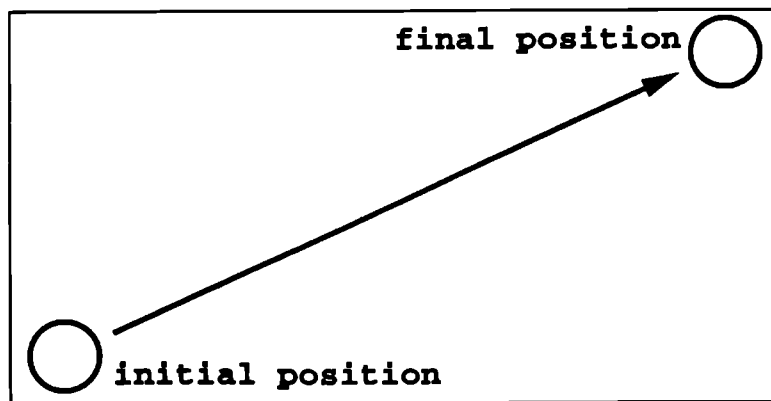


Figure 3.33: Setup for the transport of the bubble.

The “bubble” is transported across a 200×100 CV mesh with the fixed Courant number of 0.2. The influence of the temporal discretisation can be seen as a distortion in the shape of the “bubble”. The limitation on the InterGamma differencing scheme is that it can resolve the interface over no less than two control volumes.

This has been taken into account in the initial setup by smearing the interface over two cells.

Fig. 3.34 gives the initial shape of the bubble. If we recall Eqn. (3.118), it can be seen that the numerical diffusion for the Euler Implicit discretisation is large where $\mathbf{U} \cdot \nabla \phi$ is large. Fig. 3.35 illustrates the behaviour of this error term. The bubble is smeared in the direction of the velocity. In the direction normal to the flow the “bubble” is squeezed because of the conservative form of discretised equation.

The opposite effect for the Explicit discretisation, Fig. 3.36, is the result of negative numerical diffusion. The Crank-Nicholson method, Fig. 3.37 shows no distortion of the shape.

3.7.4 Comparison of Non-Orthogonality Treatments

Finally, a comparison of the three procedures for handling non-orthogonality in the solution algorithm will be given. If we limit ourselves to simple situations with analytical solutions, it is very difficult to present their performance in terms of accuracy and boundedness. It is, however, relatively easy to examine the first three criteria laid down in Section 3.5. The test case used for this purpose is a Laplace equation:

$$\nabla \cdot \nabla \phi = 0, \quad (3.135)$$

with fixed value boundary conditions on a mesh with uniform non-orthogonality shown in Fig. 3.38.

The equation is solved on 5 meshes with different non-orthogonality angles, ranging from $\alpha_n = 10^\circ$ up to $\alpha_n = 65^\circ$.

For each of the non-orthogonality treatments, the convergence history through a series of correctors for a selected computational point is plotted against the number of non-orthogonal correctors.

For the mesh angle of $\alpha_n = 10^\circ$, there is no significant difference in convergence. The overshoot in the first corrector is smallest for the over-relaxed approach.

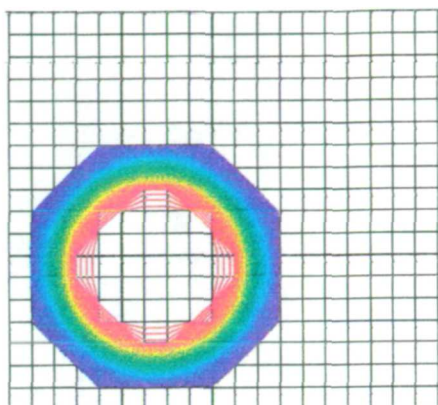


Figure 3.34: Initial shape of the bubble.

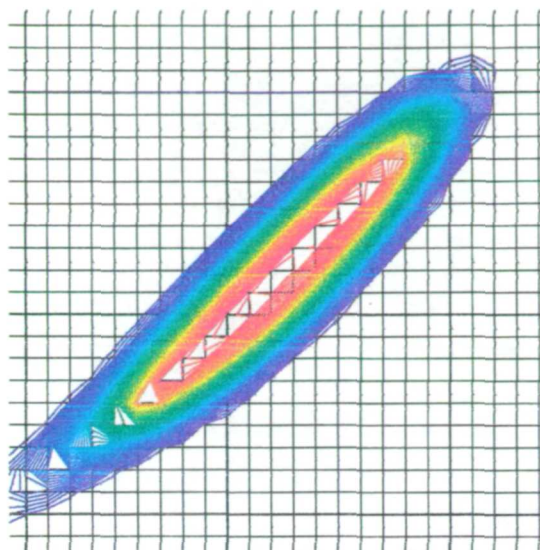


Figure 3.35: Transport of the bubble after 800 time-steps, Euler Implicit.

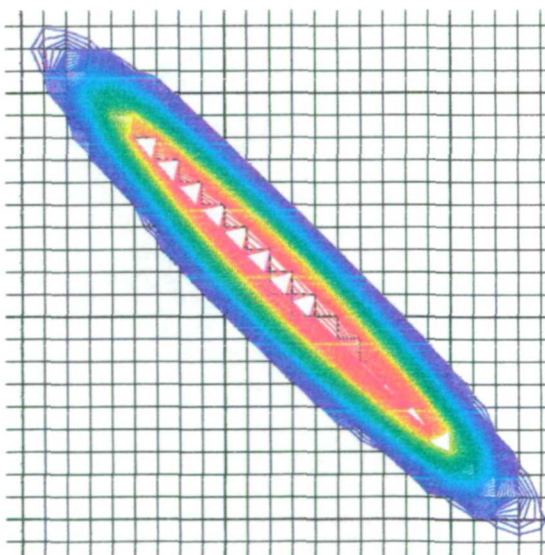


Figure 3.36: Transport of the bubble after 800 time-steps, Explicit discretisation.

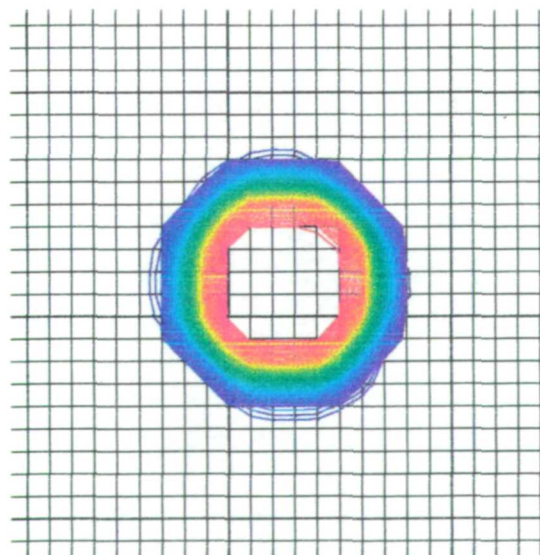


Figure 3.37: Transport of the bubble after 800 time-steps, Crank-Nicholson.

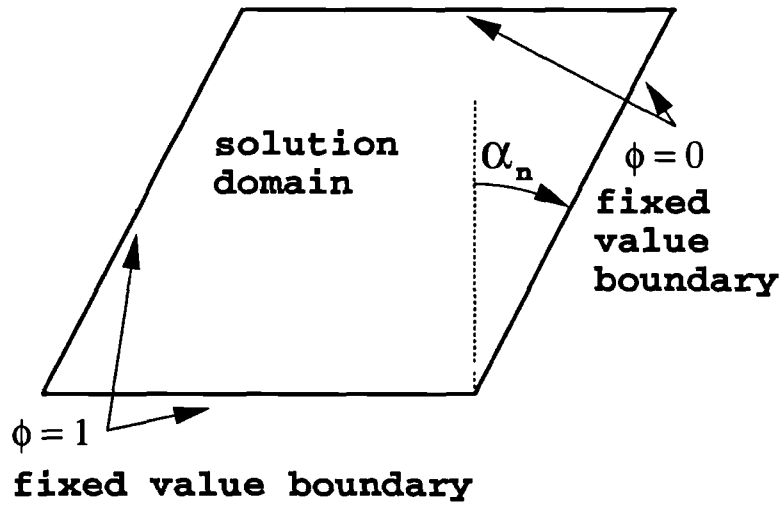


Figure 3.38: Non-orthogonal test with uniform grid angle.

For $\alpha_n = 30^\circ$, Fig. 3.40, the oscillations for the minimum correction approach are quite considerable – it is necessary to do at least five correctors to obtain a good approximation of the final solution. The over-relaxed approach needs only two correctors to obtain the solution of the same accuracy. It should also be noted that, unlike the other two approaches, its convergence curve is smooth.

When the angle is increased to 40° , Fig. 3.41, the minimum correction approach diverges (note that the scale for ϕ is much larger than on other figures). The convergence history for other two treatments is similar to the previous Figure. If the angle is increased further, Fig. 3.42, the orthogonal correction does not converge any more, even after 30 correctors. Any further increase of the angle leads to divergence.

The over-relaxed approach still converges even if the angle is increased up to 65° , Fig. 3.43. The number of non-orthogonal correctors increases considerably, as does the effort required for the solution. This should be kept in mind during the mesh generation process, specially for the transient calculations. The necessary computational effort can increase as much as an order of magnitude if mesh non-orthogonality is severe.

From these tests, it can be concluded that the over-relaxed approach has several advantages:

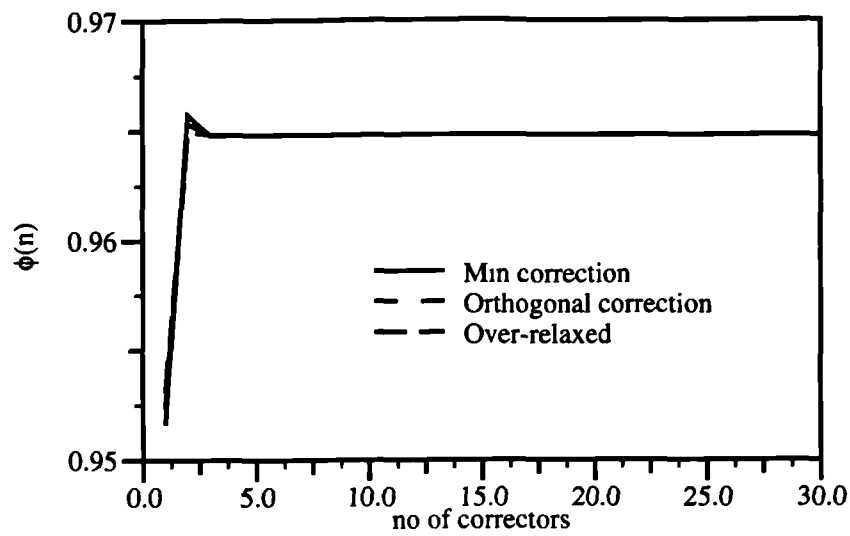


Figure 3.39: Convergence history, $\alpha_n = 10^0$.

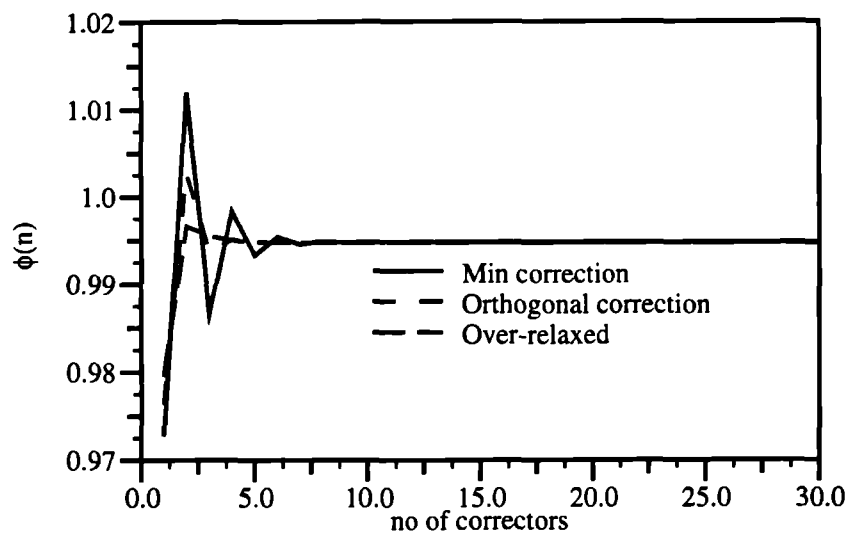


Figure 3.40: Convergence history, $\alpha_n = 30^0$.

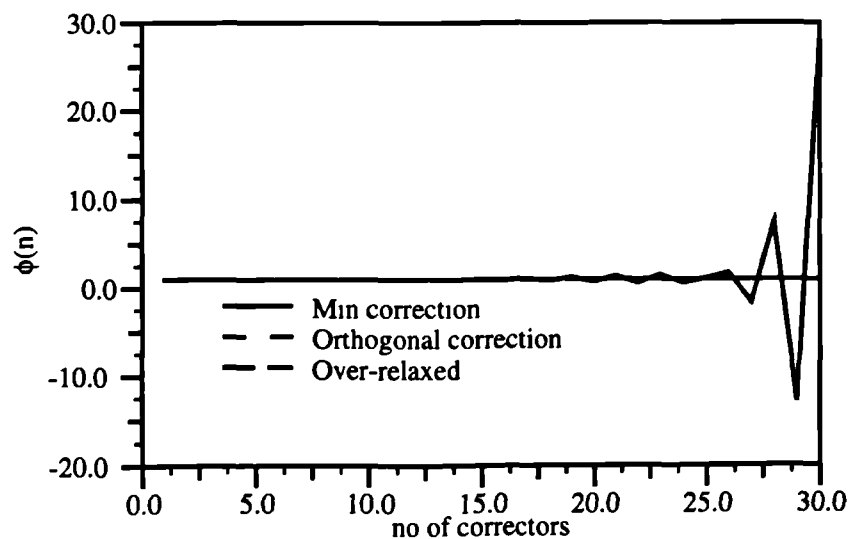


Figure 3.41: Convergence history, $\alpha_n = 40^0$.

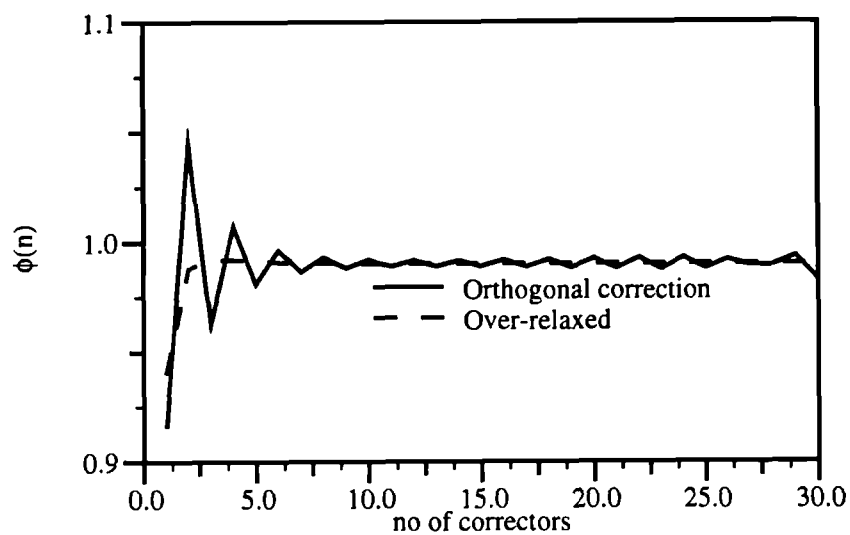


Figure 3.42: Convergence history, $\alpha_n = 45^\circ$.

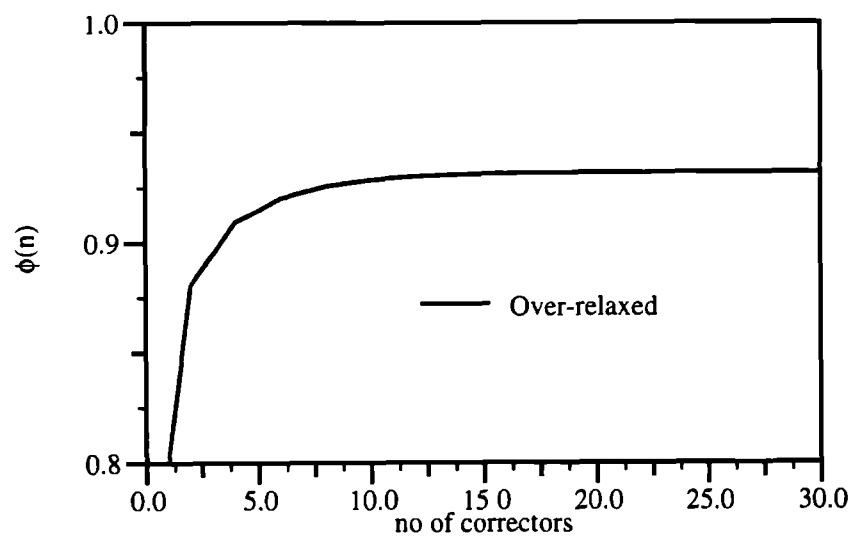


Figure 3.43: Convergence history, $\alpha_n = 65^\circ$.

- The solution after the first non-orthogonal corrector is closer to the converged solution than in other non-orthogonality treatments. The angle of mesh non-orthogonality that requires non-orthogonal correctors is therefore larger than in other approaches.
- The convergence of non-orthogonal correctors is monotonic rather than oscillatory, resulting in the higher stability of the solution procedure.
- The over-relaxed approach reaches a converged solution even on meshes distorted so severely that other non-orthogonality treatments diverge.
- It has been noted that the number of ICCG solver sweeps needed to solve a non-orthogonal corrector decreases through the corrector series, which is not the case for other non-orthogonality treatments. The overall computational cost for the over-relaxed approach will therefore be smaller even in the case of the same number of correctors.

These properties of the over-relaxed non-orthogonality treatment make it very attractive, specially for transient calculations and it will be used in the rest of this study.

3.8 Discretisation Procedure for the Navier-Stokes System

In this Section, a discretisation procedure for the Navier-Stokes equations will be presented. We shall start with the incompressible form of the system, Eqs. (2.23) and (2.24):

$$\begin{aligned}\nabla \cdot \mathbf{U} &= 0, \\ \frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{U}\mathbf{U}) - \nabla \cdot (\nu \nabla \mathbf{U}) &= -\nabla p.\end{aligned}$$

Two issues require special attention: non-linearity of the momentum equation and the pressure-velocity coupling.

The non-linear term in Eqn. (2.24) is $\nabla \cdot (\mathbf{U}\mathbf{U})$, *i.e.* velocity is “being transported by itself”. The discretised form of this expression would be quadratic in velocity and the resulting system of algebraic equations would therefore be non-linear. There are two possible solutions for this problem – either use a solver for non-linear systems, or linearise the convection term. Section 3.3.1 describes the discretisation of this term:

$$\begin{aligned}\nabla \cdot (\mathbf{U}\mathbf{U}) &= \sum_f \mathbf{S}(\mathbf{U})_f (\mathbf{U})_f \\ &= \sum_f F(\mathbf{U})_f \\ &= a_P \mathbf{U}_P + \sum_N a_N \mathbf{U}_N,\end{aligned}$$

where F , a_P and a_N are a function of $\mathbf{U}$. The important issue is that the fluxes F should satisfy the continuity equation, Eqn. (2.23). Eqs. (2.23) and (2.24) should therefore be solved together, resulting in an even larger (non-linear) system. Having in mind the complexity of non-linear equation solvers and the computational effort required, linearisation of the term is preferred. Linearisation of the convection term implies that an existing velocity (flux) field that satisfies Eqn. (2.23) will be used to calculate a_P and a_N .

The linearisation does not have any effect in steady-state calculations. When the steady-state is reached, the fact that a part of the non-linear term has been lagged is not significant. In transient flows two different approaches can be adopted: either to iterate over non-linear terms or to neglect the lagged non-linearity effects. Iteration can significantly increase the computational cost, but only if the time-step is large. The advantage is that the non-linear system is fully resolved for every time-step, whose size limitation comes only from the temporal accuracy requirements. If it is necessary to resolve the temporal behaviour well, a small time-step is needed. On the other hand, if the time-step is small, the change between consecutive solutions will also be small and it is therefore possible to lag the non-linearity without any significant effect.

In this study, the PISO procedure proposed by Issa [66] is used for pressure-

velocity coupling in transient calculations. For steady-state calculations, a SIMPLE pressure-velocity coupling procedure by Patankar [105] is used.

In Section 3.8.1 the problem of pressure-velocity coupling is presented. The pressure equation is derived for the incompressible Navier-Stokes system. Generalisation to compressible and transonic flows can be found in *eg.* Demirdzić *et al.* [39]. Section 3.8.2 describes the pressure-velocity coupling algorithms. Finally, in Section 3.8.3, a solution procedure for incompressible Navier-Stokes equations with a turbulence model is presented.

3.8.1 Derivation of the Pressure Equation

In order to derive the pressure equation, a semi-discretised form of the momentum equation will be used:

$$a_P \mathbf{U}_P = \mathbf{H}(\mathbf{U}) - \nabla p. \quad (3.136)$$

In the spirit of the Rhie and Chow procedure [117], the pressure gradient term is not discretised at this stage. Eqn. (3.136) is obtained from the integral form of the momentum equation, using the discretisation procedure described previously. It has been consequently divided through by the volume in order to enable face interpolation of the coefficients.

The $\mathbf{H}(\mathbf{U})$ term consists of two parts: the “transport part”, including the matrix coefficients for all neighbours multiplied by corresponding velocities and the “source part” including the source part of the transient term and all other source terms apart from the pressure gradient (in our case, there are no additional source terms):

$$\mathbf{H}(\mathbf{U}) = - \sum_N a_N \mathbf{U}_N + \frac{\mathbf{U}^o}{\Delta t}. \quad (3.137)$$

The discretised form of the continuity equation (Eqn. (2.23)) is:

$$\nabla \cdot \mathbf{U} = \sum_f \mathbf{S} \cdot \mathbf{U}_f = 0. \quad (3.138)$$

Eqn. (3.136) is used to express $\mathbf{U}$:

$$\mathbf{U}_P = \frac{\mathbf{H}(\mathbf{U})}{a_P} - \frac{1}{a_P} \nabla p. \quad (3.139)$$

Velocities on the cell face are expressed as the face interpolate of Eqn. (3.139 :

$$\mathbf{U}_f = \left(\frac{\mathbf{H}(\mathbf{U})}{a_P} \right)_f - \left(\frac{1}{a_P} \right)_f (\nabla p)_f. \quad (3.140)$$

This will be used later to calculate the face fluxes.

When Eqn. (3.140) is substituted into Eqn. (3.138), the following form of the pressure equation is obtained:

$$\begin{aligned} \nabla \cdot \left(\frac{1}{a_P} \nabla p \right) &= \nabla \cdot \left(\frac{\mathbf{H}(\mathbf{U})}{a_P} \right) \\ &= \sum_f \mathbf{S} \cdot \left(\frac{\mathbf{H}(\mathbf{U})}{a_P} \right)_f. \end{aligned} \quad (3.141)$$

The Laplacian on the l.h.s. of Eqn. (3.141) is discretised in the standard way (see Section 3.3.1.3).

The final form of the discretised incompressible Navier-Stokes system is:

$$a_P \mathbf{U}_P = \mathbf{H}(\mathbf{U}) - \sum_f \mathbf{S} p_f, \quad (3.142)$$

$$\sum_f \mathbf{S} \cdot \left[\left(\frac{1}{a_P} \right)_f (\nabla p)_f \right] = \sum_f \mathbf{S} \cdot \left(\frac{\mathbf{H}(\mathbf{U})}{a_P} \right)_f. \quad (3.143)$$

The face flux F is calculated using Eqn. (3.140):

$$F = \mathbf{S} \cdot \mathbf{U}_f - \mathbf{S} \cdot \left[\left(\frac{\mathbf{H}(\mathbf{U})}{a_P} \right)_f - \left(\frac{1}{a_P} \right)_f (\nabla p)_f \right]. \quad 3.144$$

When Eqn. (3.141) is satisfied, the face fluxes are guaranteed to be conservative.

3.8.2 Pressure-Velocity Coupling

Consider the discretised form of the Navier-Stokes system, Eqs. 3.142) and (3.143). The form of the equations shows linear dependence of velocity on pressure and vice-versa. This inter-equation coupling requires a special treatment.

Simultaneous algorithms (eg. Caretto *et al.* [23], Vanka [143]) operate by solving the complete system of equations simultaneously over the whole domain. Such a procedure might be considered when the number of computational points is small and the number of simultaneous equations is not too large. The resulting

matrix includes the inter-equation coupling and is several times larger than the number of computational points. The cost of a simultaneous solution is great, both in the number of operations and memory requirements.

In the **segregated approach** (eg. Patankar [105], Issa [66]) the equations are solved in sequence. A special treatment is required in order to establish the necessary inter-equation coupling. PISO [66], SIMPLE [105] and their derivatives are the most popular methods of dealing with inter-equation coupling in the pressure-velocity system.

3.8.2.1 The PISO Algorithm for Transient Flows

This pressure-velocity treatment for transient flow calculations has been originally proposed by Issa [66]. Let us again consider the discretised Navier-Stokes system for incompressible flow, Eqs. (3.142) and (3.143). The PISO algorithm can be described as follows:

- The momentum equation is solved first. The exact pressure gradient source term is not known at this stage – the pressure field from the previous time-step is used instead. This stage is called the **momentum predictor**. The solution of the momentum equation, Eqn. (3.142), gives an approximation of the new velocity field.
- Using the predicted velocities, the $\mathbf{H} \mathbf{U}$ operator can be assembled and the pressure equation can be formulated. The solution of the pressure equation gives the first estimate of the new pressure field. This step is called the **pressure solution**.
- Eqn. (3.144) gives a set of conservative fluxes consistent with the new pressure field. The velocity field should also be corrected as a consequence of the new pressure distribution. Velocity correction is done in an explicit manner, using Eqn. (3.139). This is the **explicit velocity correction** stage.

A closer look to Eqn. (3.139) reveals that the velocity correction actually consists of two parts – a correction due to the change in the pressure gradient ($\frac{1}{\alpha_p} \nabla p$ term)

and the transported influence of corrections of neighbouring velocities ($\frac{\mathbf{H}(\mathbf{U})}{a_P}$ term). The fact that the velocity correction is explicit means that the latter part is neglected – it is effectively assumed that the whole velocity error comes from the error in the pressure term. This, of course, is not true. It is therefore necessary to correct the $\mathbf{H}(\mathbf{U})$ term, formulate the new pressure equation and repeat the procedure. In other words, the PISO loop consists of an implicit momentum predictor followed by a series of pressure solutions and explicit velocity corrections. The loop is repeated until a pre-determined tolerance is reached.

Another issue is the dependence of $\mathbf{H}(\mathbf{U})$ coefficients on the flux field. After each pressure solution, a new set of conservative fluxes is available. It would be therefore possible to recalculate the coefficients in $\mathbf{H}(\mathbf{U})$. This, however, is not done: it is assumed that the non-linear coupling is less important than the pressure-velocity coupling, consistent with the linearisation of the momentum equation. The coefficients in $\mathbf{H}(\mathbf{U})$ are therefore kept constant through the whole correction sequence and will be changed only in the next momentum predictor.

3.8.2.2 The SIMPLE Algorithm

If a steady-state problem is being solved iteratively, it is not necessary to fully resolve the linear pressure-velocity coupling, as the changes between consecutive solutions are no longer small. Non-linearity of the system becomes more important, since the effective time-step is much larger.

The SIMPLE algorithm by Patankar [105] is formulated to take advantage of these facts:

- An approximation of the velocity field is obtained by solving the momentum equation. The pressure gradient term is calculated using the pressure distribution from the previous iteration or an initial guess. The equation is under-relaxed in an implicit manner (see Eqn. (3.96)), with the **velocity under-relaxation factor** α_U .
- The pressure equation is formulated and solved in order to obtain the new

pressure distribution.

- A new set of conservative fluxes is calculated using Eqn. (3.144). As it has been noticed before, the new pressure field includes both the pressure error and convection-diffusion error. In order to obtain a better approximation of the “correct” pressure field, it would be necessary to solve the pressure equation again. On the other hand, the non - linear effects are more important than in the case of transient calculations. It is enough to obtain an approximation of the pressure field and recalculate the $\mathbf{H}(\mathbf{U})$ coefficients with the new set of conservative fluxes. The pressure solution is therefore **under-relaxed** in order to take into account the velocity part of the error:

$$p^{new} = p^{old} + \alpha_p(p^p - p^{old}) \quad (3.145)$$

where

- p^{new} is the approximation of the pressure field that will be used in the next momentum predictor,
- p^{old} is the pressure field used in the momentum predictor,
- p^p is the solution of the pressure equation,
- α_p is the **pressure under-relaxation factor**, ($0 < \alpha_p \leq 1$).

If the velocities are needed before the next momentum solution, the explicit velocity correction, Eqn. (3.139), is performed.

Perić, [109] gives an analysis of the under-relaxation procedure based on the expected behaviour of the second corrector in the PISO sequence. The recommended values of under-relaxation factors are (Perić, [109]):

- $\alpha_p = 0.2$ for the pressure and
- $\alpha_U = 0.8$ for momentum.

3.8.3 Solution Procedure for the Navier-Stokes System

It is now possible to describe the solution sequence for the Navier-Stokes system with additional coupled transport equations (*eg.* a turbulence model, combustion equations, energy equation or some other equations that influence the system).

In transient calculations, all inter-equation couplings apart from the pressure-velocity system are lagged. If it is necessary to ensure a closer coupling between some of the equations (*eg.* energy and pressure in combustion), they are included in the PISO loop. A **transient solution procedure** for incompressible turbulent flows can be summarised as follows:

1. Set up the initial conditions for all field values.
2. Start the calculation of the new time-step values.
3. Assemble and solve the momentum predictor equation with the available face fluxes.
4. Go through the PISO loop until the tolerance for pressure-velocity system is reached. At this stage, pressure and velocity fields for the current time-step are obtained, as well as the new set of conservative fluxes.
5. Using the conservative fluxes, solve all other equations in the system. If the flow is turbulent, calculate the effective viscosity from the turbulence variables.
6. If the final time is not reached, return to step 2.

The solution procedure for **steady-state incompressible turbulent flow** is similar:

1. Set all field values to some initial guess.
2. Assemble and solve the under-relaxed momentum predictor equation
3. Solve the pressure equation and calculate the conservative fluxes. Update the pressure field with an appropriate under-relaxation. Perform the explicit velocity correction using Eqn. (3.139)

4. Solve the other equations in the system using the available fluxes, pressure and velocity fields. In order to improve convergence, under-relax the other equations in an implicit manner, as shown in Eqn. (3.96).
5. Check the convergence criterion for all equations. If the system is not converged, start a new iteration on step 2.

3.9 Closure

A Finite Volume discretisation technique for arbitrarily unstructured meshes has been described. It allows the use of the control volumes of arbitrary topology, simplifying the problem of mesh generation for complex geometries. In order to preserve the efficiency of the algorithm, a “face addressing” approach by Weller [147] has been adopted.

The discretisation of the temporal and spatial terms based on the face addressing procedure has been described in Section 3.3. The proposed discretisation treatment is second-order accurate in space and time. Several methods of temporal discretisations have also been examined. Implementation of boundary conditions has been briefly discussed. A new non-orthogonality treatment has been proposed and tested on a series of meshes with increasing non-orthogonality angles. The results show that the over-relaxed approach has distinct advantages over the other two approaches in terms of stability, although it causes the largest discretisation error if the non-orthogonal correction is discarded. It provides better convergence behaviour and at the same time allows higher angles of non-orthogonality.

In order to improve the discretisation of the convection term, the Gamma differencing scheme has been proposed in Section 3.4. It is based on the Normalised Variable Approach, first suggested by Leonard [82]. It guarantees bounded solutions and preserves the second-order accuracy of the method. The Gamma differencing scheme uses a compact computational molecule (including only first neighbours of the control volume), making it specially useful on arbitrarily unstructured meshes. A necessary modification in the formulation of the Normalised Variable is also pre-

sented.

Particular attention has been paid to the order of accuracy of the method and the sources of the discretisation error. The main sources of the error are the convection differencing scheme, discretisation of the transient terms and the mesh quality. The effective numerical diffusion has been derived for each of those terms. Numerical examples, showing the influence of different forms of numerical diffusion to the solution, have been presented in Section 3.7.

Finally, in Section 3.8, a discretisation procedure for coupled systems of equations has been presented. The adopted treatment of the pressure-velocity system is based on the PISO algorithm for transient calculations and the SIMPLE approach for steady-state flows. A sequence of operations for both steady-state and transient calculations has been summarised.

Chapter 4

Error Estimation

4.1 Introduction

Chapter 3 describes the Finite Volume discretisation. The discretisation procedure allows us to obtain a discrete approximation of governing equations for a particular problem. When the resulting system of algebraic equations is solved, it produces an approximate solution at a set of points in the computational domain. The quality of the numerical solution depends on the distribution of the points as well as the applied discretisation practice.

The presented method of discretisation is second-order accurate in space and time, *i.e.* it is assumed that the variation of the function over each control volume is linear. This is the major source of numerical errors. If a better solution is needed, control volumes should be subdivided in such a way that the assumption about the linear variation becomes acceptable.

Another source of numerical errors is the discretisation practice. For good accuracy, it is necessary for the order of discretisation to be equal to or higher than the order of the partial differential equation that is being discretised. If that is not the case, discretisation errors effectively introduce a term which is of the order of other terms in the equation, resulting in high numerical errors unless the mesh is made excessively fine.

The error in the numerical solution is defined as the difference between the exact

solution of the governing equation $\Phi(\mathbf{x}, t)$, and the solution of the discrete system ϕ :

$$E = \Phi - \phi. \quad (4.1)$$

The purpose of error estimation is to detect inaccuracies in the numerical solution, including both the errors coming from inappropriate discretisation of the solution domain and discretisation errors. Adaptive refinement process consists of a series of numerical solutions and error estimations, followed by appropriate improvements in the discretisation. The objective of the procedure is to automatically produce a numerical solution of the prescribed accuracy.

The role of the error estimate is to provide an answer to the following questions:

- What is the magnitude of the solution error?
- If the error is larger than some pre-determined level, what is the most effective way of reducing it?

The answer to the first question controls the adaptive refinement process. If the current solution satisfies the error level criterion, further refinement is no longer necessary. If this is not the case, the error distribution can be used to introduce some changes in the numerical procedure *eg.* better distribution of computational points or higher order of discretisation, in order to improve the accuracy of consequent solutions.

The only data available to the analyst containing the information about the error is the numerical solution itself. This leads us to the concept of ***a-posteriori* error estimation**. As the name suggests, an estimate of the solution error is obtained after the numerical solution. An *a-posteriori* error estimate is constructed having in mind the known characteristics of the discretisation practice and the equation that is being solved.

4.1.1 Error Estimators and Error Indicators

Early efforts in the field of error detection have been directed towards problems with discontinuous solutions, like multiple shock structures in supersonic flows (see Berger and Oliger [17], Berger and Collela [16], Löhner [89], Ramakrishnan *et al.* [116] *etc.*). The solution has some distinct features – shocks, contact discontinuities, shock-to-shock interaction points *etc.* that need to be resolved accurately. It is known in advance that those features can be recognised by large gradients in flow variables. Adaptive refinement is therefore directed towards the regions in which gradients are high. In the exact solution, discontinuities are infinitely thin – mesh refinement is therefore stopped when the thickness of discontinuities in the numerical solution is considered to be small enough, or when the maximum number of computational points is reached.

Combinations of gradients of flow variables used to control the adaptive refinement procedure are called **error indicators**. An error indicator highlights the regions of the domain where a better resolution is needed. In general, it does not provide information about the absolute error level. Error indicators are cheap to compute and for simple situations give reliable information about the solution.

Adaptive refinement for problems with smooth solutions cannot be controlled by an error indicator. Flow features that need refinement cannot be identified in advance and adaptive refinement should be stopped when some pre-determined accuracy is reached. An **error estimate** is therefore expected to give more information about the accuracy than the error indicator. It should estimate the absolute error level as well as the distribution of the error throughout the domain. Interaction between error estimation and local mesh refinement should also be kept in mind. If the refinement algorithm allows localised refinement with rapid changes in mesh size, it is necessary to obtain accurate information about the error. Patch-based refinement methods are not so sensitive to the accuracy of the error estimate.

The **Effectivity index** ζ measures the quality of an error estimate. It is defined

as the ratio of magnitudes of the estimated error e and the exact error E :

$$\zeta = \frac{e}{E}. \quad (4.2)$$

Good error estimates have effectivity indices close to unity.

In this Chapter several methods of error estimation are proposed. Section 4.2 outlines the requirements on an error estimate. In order to simplify matters, we shall first examine the problem of error estimation in steady-state calculations, where the whole error comes from the discretisation of spatial terms. Three approaches to *a-posteriori* error estimation of the spatial error are examined.

The first group of methods derive the error from the known order of accuracy and the local variation of the solution. They are based on the Taylor series truncation error analysis, discussed in Section 4.3. Two different ways of estimating the first term of the truncation error are presented: Richardson extrapolation, Section 4.3.1 and the Direct Taylor Series Error estimate. Section 4.3.2.

Section 4.4 describes a new approach to the problem of error estimation – the Moment Error estimate. It is derived from the analysis of the original equation in the differential form. The imbalance in the transport equation for a higher moment of the variable is used to estimate the error. Instead of measuring the numerical error directly, it derives its value from the effects of the discretisation inaccuracy on the balance in the discretised moment equation.

An alternative approach to error estimation is presented in Section 4.5. The Residual Error estimate follows similar work in Finite Element error estimation. Residual is a function that measures how well the original equations are satisfied over each control volume. It is assembled on a cell-by-cell basis. A transport-based normalisation of cell residuals is used to estimate the absolute error level.

Section 4.6 gives an extension of the Residual Error estimate. It is possible to derive a strict upper bound on the solution error expressed through an appropriate error norm. Unlike the other error estimation methods described in this Chapter, it does not measure the error directly. The Local Problem Error estimate is based on the similar work of Bank and Weiser [12], Kelly [71], Oden *et al.* [98], Ainsworth

and Oden [1, 2, 3, 4] and others in the Finite Element community. The approach by Ainsworth and Oden [4] has been chosen as the most general and directly extendible to the Finite Volume method. The basic error estimate has been developed for linear elliptic problems and consequently extended to unisymmetric [2] and non-linear (specifically incompressible Navier-Stokes equations) [101] problems. The error is presented in the form of an error norm. One of the requirements for the successful use of this error estimate is an accurate error flux balancing procedure. This has been widely discussed in the Finite Element literature (see Kelly [71], Ainsworth and Oden [3] or Ainsworth [4]). The conservation properties of the Finite Volume solution removes the need for a special error flux balancing algorithm, as will be shown in Section 4.6.1.1. A simplified solution procedure for the local error problem is given in Section 4.6.4.

Error estimation for transient calculations is discussed in Section 4.7. The transient error estimate is derived as an extension of the Residual method, taking into account the effects of the temporal discretisation on the accuracy of the solution. It is possible to split the total error into the spatial and temporal contributions. The requirement of equivalent temporal and spatial accuracy can be used to determine the optimal time-step size for a given computational mesh.

The examples presented in Section 4.8 illustrate the performance of error estimates in simple situations. Comparison of the estimated and real error is given for test cases with analytical solutions. Finally, some closing remarks are given in Section 4.9.

4.2 Requirements on an Error Estimate

Let us now outline a set of properties a good error estimate should have.

The purpose of error estimation is to give an insight into the accuracy of the solution. It is desirable to present the error information in terms of the **absolute error value**. The error estimate is easier to understand if its dimensions are the same as those of the solution variable. This approach is not generally accepted

because of its impracticality for singular problems. A typical example can be found in linear elasticity (see Zienkiewicz [156]). Under a point load, the local displacement (and hence the error) is locally equal to infinity. The net effect of this error on the rest of the domain is usually negligible (Zienkiewicz [156]). The solution error for such situations can, however, be presented as an error norm. Singular problems are not common in fluid flows and such treatment is not necessary.

The error estimate will be used to control an adaptive refinement procedure. It should therefore provide **reliable information about the distribution of the error** through the computational domain. Even if the error level is not estimated accurately, a correct error distribution guarantees that refinement regions are properly selected.

Accurate error distribution is critical in calculations with highly localised refinement, where the successive levels of refinement are embedded into each other. First levels of refinement should be located accurately in order to reduce unnecessary overheads. Error estimation should therefore **work well on coarse meshes**.

The error estimate will regularly be calculated on adaptively refined meshes, consisting of control volumes of different sizes. It is even possible to use different orders of discretisation in different parts of the mesh. If an accurate estimate of the error is to be obtained for such a mesh, the error estimate should **scale correctly with the order of discretisation and mesh size**.

For transient calculations with tracking of flow features, error estimation will have to be done very often. The mesh is refined every few time-steps, depending on the level of the error. In order to keep the computational cost at a reasonable level, both error estimation and mesh refinement need to be computationally efficient. The error estimate should therefore be **easy to compute**. An efficient error estimate is **based on the local solution and mesh information** and can be calculated on a cell-by-cell basis. If error estimation requires a solution of a system of algebraic equations over all control volumes, it is considered to be too expensive.

Error estimation method should have a **strong mathematical basis**. It should be applicable to all degenerate forms of the transport equation (elliptic problems,

convection or source-dominated problems). It is also desirable that the error estimate gives the exact value of the error if the order of solution is one order higher than the order of discretisation.

The transient error estimate is expected to work well even if the change of the solution in time is very slow, or the solution tends to a steady-state solution. It would therefore be advisable to construct it as an extension of a steady-state error estimator.

As the number of computational points goes to infinity, the error estimate should tend to the exact error faster than the solution tends to the exact solution. The error estimate with this property is said to be **asymptotically exact**. It is expected that the error estimate will detect both the errors resulting from insufficient mesh resolution and the discretisation errors. The error estimate should also be **reliable** in the vicinity of points of singularity.

Finally, the error estimate should modestly **over-estimate the actual error**. Considering the complexity of error estimation, one cannot expect extremely accurate results. If it is known that the error can be estimated only up to a certain accuracy, over-estimation is preferred. Once the desired accuracy of the solution is achieved according to the error estimate, the actual error will be even smaller.

Some of the aforementioned requirements on the error estimate are quite hard to fulfill, others are even contradictory. It would be difficult to create an error estimate which is both very accurate and easy to compute. Accuracy of the estimate on coarse meshes presents a particular problem. The error estimation methods presented in this Chapter should be seen as a compromise between the suggested requirements.

4.3 Methods Based on Taylor Series Expansion

The first group of error estimation methods is based on the analysis of the numerical solution in terms of the Taylor series expansion. Every smooth function can be written as an expansion in its derivatives around a given point in space. This expansion is potentially infinite, depending on the number of non-zero derivatives

at the selected point.

The discretisation process can be considered as a truncation of the infinite series. The truncated form of the expansion at the computational point is used to describe the variation of the solution over the control volume surrounding it. The complete solution is created as a union of these locally defined “shape functions”.

A p th order accurate discretisation method describes the local variation of ϕ with the first p terms of the Taylor series:

$$\phi(\mathbf{x}) = \phi_P + (\mathbf{x} - \mathbf{x}_P) \cdot (\nabla \phi)_P + \dots + \frac{1}{(p-1)!} (\mathbf{x} - \mathbf{x}_P)^{p-1} \underbrace{\underbrace{\nabla \nabla \dots \nabla}_{p-1} \phi}_P. \quad (4.3)$$

The discretisation error can also be expressed as an (infinite) series in higher derivatives of ϕ :

$$e(\mathbf{x}) = \Phi(\mathbf{x}) - \phi(\mathbf{x}) = \sum_{n=p}^{\infty} \frac{1}{n!} (\mathbf{x} - \mathbf{x}_P)^n \underbrace{\underbrace{\nabla \nabla \dots \nabla}_n \phi}_P. \quad (4.4)$$

The estimate of the error in the computational point is based on the average error in the control volume, consistent with the discretisation practice:

$$\begin{aligned} e_t(\phi) &= \left| \frac{1}{V_P} \int_{V_P} \left[\sum_{n=p}^{\infty} \frac{1}{n!} (\mathbf{x} - \mathbf{x}_P)^n \underbrace{\underbrace{\nabla \nabla \dots \nabla}_n \phi}_P \right] dV \right| \\ &\leq \frac{1}{V_P} \sum_{n=p}^{\infty} \left| \int_{V_P} \left(\frac{1}{n!} (\mathbf{x} - \mathbf{x}_P)^n \underbrace{\underbrace{\nabla \nabla \dots \nabla}_n \phi}_P \right) dV \right|. \end{aligned} \quad (4.5)$$

Let us now consider the form of Eqn. (4.5). It is assumed that the magnitude of the leading term in Eqn. (4.5) gives a good estimate of the whole truncation error. If the exact solution $\Phi(\mathbf{x})$ is smooth and the control volume is small, contributions from higher terms of the expansion in Eqn. (4.5) will indeed decrease rapidly with increasing n . This is strictly true only if the mesh resolution is adequate.

The Taylor series error estimate uses only the first term of the expansion in Eqn. 4.5) to estimate the error:

$$\begin{aligned} e_t(\phi) &= \frac{1}{V_P} \int_{V_P} \left(\frac{1}{p!} (\mathbf{x} - \mathbf{x}_P)^p \underbrace{\underbrace{\nabla \nabla \dots \nabla}_p \phi}_P \right) dV \\ &\quad - \frac{1}{V_P} \frac{1}{p!} \left[\int_{V_P} (\mathbf{x} - \mathbf{x}_P)^p dV \right] \underbrace{\underbrace{\nabla \nabla \dots \nabla}_p \phi}_P. \end{aligned} \quad (4.6)$$

If the mesh is too coarse, the contribution of higher-order terms can be significant, particularly if higher derivatives are large. It can therefore be expected that Eqn. (4.6) under-estimates the actual error.

In the case of second order accurate (Finite Volume) discretisation, the prescribed spatial variation of ϕ over the control volume is linear, Eqn. (3.3):

$$\phi(\mathbf{x}) = \phi_P + (\mathbf{x} - \mathbf{x}_P) \cdot (\nabla \phi)_P$$

The leading term of the truncation error is:

$$e(\mathbf{x}) = \frac{1}{2}(\mathbf{x} - \mathbf{x}_P)^2 : (\nabla \nabla \phi)_P. \quad (4.7)$$

Following Eqn. (4.6), the Taylor series error estimate in the point P is calculated as:

$$\begin{aligned} e_t(\phi) &= \frac{1}{V_P} \frac{1}{2} \left| \left[\int_{V_P} (\mathbf{x} - \mathbf{x}_P)^2 dV \right] : (\nabla \nabla \phi)_P \right| \\ &= \frac{1}{2 V_P} \mathbf{M} : (\nabla \nabla \phi)_P. \end{aligned} \quad (4.8)$$

$\mathbf{M}$ in Eqn. (4.8) is the second geometric moment of the control volume:

$$\mathbf{M} = \int_{V_P} (\mathbf{x} - \mathbf{x}_P)^2 dV. \quad (4.9)$$

The local error can be estimated in two ways, resulting in two methods of error estimation. The first, Richardson extrapolation, uses numerical solutions from two meshes with different spacing. The second method estimates the numerical error from a single solution. An error estimate based on the latter idea is presented in Section 4.3.2 below.

4.3.1 Richardson Extrapolation

Richardson extrapolation is the most popular form of error estimation based on the Taylor series truncation error analysis. It has been used for a large variety of flow problems, ranging from incompressible Navier-Stokes equations (see *eg.* Thompson and Ferziger [134], Muzaferija [97], Chen *et al.* [28]) to inviscid supersonic flows (see

eg. Berger and Oliger [17], Berger and Collela [16]). Since the method requires solutions of the same problem on two meshes, it has been regularly used in conjunction with multigrid acceleration techniques.

The basic idea of Richardson extrapolation is to obtain an approximation of the leading term in the truncation error from suitably weighted solutions on two meshes with different cell size (Muzaferija [97], Caruso [25]).

The spatial variation of the exact solution on two meshes with spacing h_1 and h_2 can be symbolically written as (Muzaferija [97]):

$$\Phi(\mathbf{x}) = \phi(\mathbf{x}, h_1) + h_1^p C(\mathbf{x}) + O(\mathbf{x}, h_1^q), \quad (4.10)$$

$$\Phi(\mathbf{x}) = \phi(\mathbf{x}, h_2) + h_2^p C(\mathbf{x}) + O(\mathbf{x}, h_2^q), \quad (4.11)$$

where

- $\phi(\mathbf{x}, h_i)$ is the approximate solution on the mesh with spacing h_i ,
- $h_i^p C(\mathbf{x})$ is the leading term of the truncation error,
- $h = h(\mathbf{x})$ is the local mesh size calculated as the ratio of cell volume and surface area:

$$h = \frac{V_P}{\sum_f S}, \quad (4.12)$$

- p is the order of accuracy of the discretisation method,
- $O(\mathbf{x}, h_i^q)$ is the rest of the truncation error.

From Eqs. (4.10) and (4.11), $C(\mathbf{x})$ can be approximated as:

$$C(\mathbf{x}) = \frac{\phi(\mathbf{x}, h_2) - \phi(\mathbf{x}, h_1)}{h_1^p - h_2^p}. \quad (4.13)$$

The estimate of $C(\mathbf{x})$, Eqn. (4.13), can be used to improve the fine mesh solution $\phi(\mathbf{x}, h_2)$. The improved (q th order accurate) solution is:

$$\phi(\mathbf{x}, 0) = \phi(\mathbf{x}, h_2) \frac{\left(\frac{h_1}{h_2}\right)^p}{\left(\frac{h_1}{h_2}\right)^p - 1} - \phi(\mathbf{x}, h_1) \frac{1}{\left(\frac{h_1}{h_2}\right)^p - 1}. \quad (4.14)$$

This improved solution can be used to estimate the error in $\phi(\mathbf{x}, h_2)$.

The Richardson extrapolation error estimate is calculated from the difference between the improved solution and the solution from the fine mesh:

$$e_t(\phi) = |\phi(\mathbf{x}, 0) - \phi(\mathbf{x}, h_2)| = \frac{\phi(\mathbf{x}, h_2) - \phi(\mathbf{x}, h_1)}{\left(\frac{h_1}{h_2}\right)^p - 1}. \quad (4.15)$$

For second-order accurate discretisation:

$$e_t(\phi) = \frac{|\phi(\mathbf{x}, h_2) - \phi(\mathbf{x}, h_1)|}{\left(\frac{h_1}{h_2}\right)^2 - 1}. \quad (4.16)$$

It is instructive to analyse the interaction between the Richardson extrapolation error estimate, Eqn. (4.16), and the general form of the Taylor series error estimate, Eqn. (4.8). Let us start with the simplest form of the second geometric moment for a hexahedral control volume aligned with the global coordinate system, Fig. 4.1.

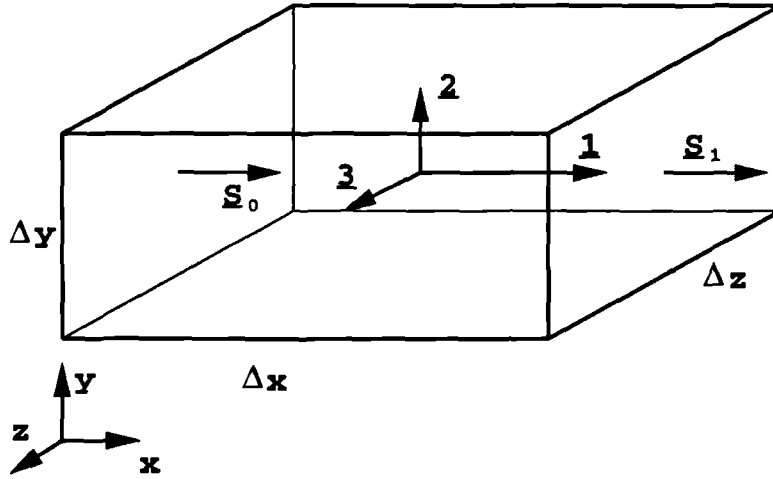


Figure 4.1: Hexahedral control volume aligned with the coordinate system.

The calculation of the moment tensor $\mathbf{M}$ is now straightforward:

$$\mathbf{M} = \frac{V_P}{12} \begin{bmatrix} \Delta x^2 & 0 & 0 \\ 0 & \Delta y^2 & 0 \\ 0 & 0 & \Delta z^2 \end{bmatrix}. \quad (4.17)$$

The Taylor series error estimate for this control volume, Eqn. (4.8), simplifies to:

$$\begin{aligned} e_t(\phi) &= \frac{1}{2 V_P} \mathbf{M} : (\nabla \nabla \phi)_P \\ &= \frac{1}{24} \left[\Delta x^2 \left(\frac{\partial^2 \phi}{\partial x^2} \right)_P + \Delta y^2 \left(\frac{\partial^2 \phi}{\partial y^2} \right)_P + \Delta z^2 \left(\frac{\partial^2 \phi}{\partial z^2} \right)_P \right]. \end{aligned} \quad (4.18)$$

It is also known that $(\nabla\nabla\phi)_P$ should be independent of the mesh, as two numerical solutions correspond to the same physical problem. The double-dot product $\mathbf{M} : (\nabla\nabla\phi)_P$ in Eqn. (4.18) is expected to scale quadratically with h . This is independent of the orientation of the control volume in relation to the global coordinate system. Richardson extrapolation, Eqn. (4.16), uses this property to estimate the error. According to Eqn. (4.18), Richardson extrapolation assumes that the magnitude of each term in $\mathbf{M}$ individually scales with h^2 . This is valid only for geometrically similar control volumes. If the control volumes of two meshes are not similar in shape, Richardson extrapolation loses accuracy.

The Richardson extrapolation error estimate requires two solutions on meshes with different spacing: $\phi(\mathbf{x}, h_1)$ and $\phi(\mathbf{x}, h_2)$. In industrial applications this is not always feasible – the mesh necessary to appropriately describe the geometry and boundary conditions can be of the order of several hundred thousands cells. Calculation on a finer mesh could easily exceed the available computational resources. The adaptive refinement algorithm proposed in this study does not include multi-grid acceleration and two solutions are not readily available. As a consequence, the Richardson extrapolation error estimate is considered to be expensive.

It is well known that Richardson extrapolation under-estimates the solution error. It is particularly inaccurate on coarse meshes. If a good error estimate is to be obtained, both meshes need to be fine enough to approximate the solution well. Once the solution is smooth in relation to mesh size on both meshes, the higher-order truncation error terms vanish quickly and the accuracy of the error estimate improves rapidly with mesh refinement.

4.3.2 Direct Taylor Series Error Estimate

It is possible to obtain the Taylor series error estimate from a single-mesh result. This is a novel approach to Taylor series error estimation. The terms from Eqn. (4.8) will be approximated from the available solution and mesh geometry. The error estimate obtained in such a way is called the **Direct Taylor Series Error estimate**.

The task of creating a single-mesh Taylor series error estimate can be divided into

two parts: estimating the second gradient of ϕ in P and evaluating the geometric moment tensor $\mathbf{M}$ for the control volume, both appearing in Eqn. (4.18).

The $(\nabla\nabla\phi)_P$ tensor is calculated from the available solution using Gauss' theorem twice: Eqn. (3.10) is used to calculate $(\nabla\phi)_P$, which is then used in Eqn. (3.11) to obtain the second gradient. It should be noted however that the double Gauss' theorem actually provides an average value of the second gradient over the control volume and the set of its neighbours, whereas Eqn. (4.8) requires the value in the centroid of the control volume. The estimate will therefore be good only if the numerical solution approximates the exact solution well and the mesh is fine enough.

Calculation of $\mathbf{M}$ for control volumes in an arbitrarily unstructured mesh is slightly more complex. The approach adopted here follows the work of Helf and Küster [62]. Using Gauss' theorem, it is possible to reduce the volume integral in Eqn. (4.9) to a set of geometric moment integrals over the faces and finally to integrals over face edges. In order to simplify the derivation, the terms of $\mathbf{M}$ will be assembled in a local coordinate system, with the origin in the centre of the control volume. Eqn. (4.9) can be modified as follows:

$$\begin{aligned}\mathbf{M} &= \int_{V_P} (\mathbf{x})^2 dV = \int_{V_P} (\mathbf{x} \mathbf{x}) dV \\ &= \frac{1}{3} \int_{V_P} \nabla \cdot [\mathbf{x} (\mathbf{x} \mathbf{x})] dV \\ &= \frac{1}{3} \int_{\partial V_P} (d\mathbf{S} \cdot \mathbf{x}) (\mathbf{x} \mathbf{x}) \\ &= \frac{1}{3} \sum_f \int_{S_f} (d\mathbf{S} \cdot \mathbf{x}) (\mathbf{x} \mathbf{x}).\end{aligned}\tag{4.19}$$

The surface integrals in Eqn. (4.19) can be further simplified. It is known that all faces are flat. For every point on the face, the dot-product $d\mathbf{S} \cdot \mathbf{x}$ reduces to a constant. It follows that:

$$\int_{S_f} (d\mathbf{S} \cdot \mathbf{x}) (\mathbf{x} \mathbf{x}) = \hat{\mathbf{S}} \cdot \mathbf{x}_f \int_{S_f} (\mathbf{x} \mathbf{x}) dA.\tag{4.20}$$

Combining Eqs. (4.19) and (4.20), the volume integral is reduced to a sum of surface integrals over flat faces. Point $\mathbf{x}_f$ has been selected as a sample point – any

other point from the face plane will give the same product. Once the face integrals are calculated for all faces of the mesh, they are added to control volumes on both sides of the face with opposite signs.

Repeating the procedure once again, calculation of second geometric moments for the face can be reduced to a sum of integrals over the face edges, which are easily calculated.

This method is directly applicable on general polyhedral control volumes. In spite of its recursive nature, it still seems computationally expensive. Calculation of the error requires the evaluation of two second rank tensors (or a second rank and a third rank tensor for vector properties), which is considered expensive.

Having in mind the limited accuracy of $(\nabla\nabla\phi)_P$, it would be wise to produce a simple estimate of $\mathbf{M}$, knowing that this simplification does not significantly degrade the overall accuracy of the method. An appropriate transformation of the coordinate system can reduce the second geometric moment tensor to diagonal terms only. Orientation of the coordinate system then coincides with the principal geometric axes of inertia for the control volume. If the product $\mathbf{M} : (\nabla\nabla\phi)_P$ were calculated in that coordinate system, it would reduce to only three terms, similar to Eqn. (4.18). Vectors **1**, **2** and **3** in Fig. 4.1 represent the principal vectors of inertia for the control volume. For a hexahedral cell of arbitrary orientation, they can be approximated from the face area vectors of the opposite face pairs. Thus, for example, **1** is approximated with (Fig. 4.1):

$$\mathbf{1} = \left(\frac{\mathbf{S}_0}{|\mathbf{S}_0|} + \frac{\mathbf{S}_1}{|\mathbf{S}_1|} \right) \frac{V_P}{|\mathbf{S}_0| + |\mathbf{S}_1|} \quad (4.21)$$

and the error estimate, Eqn. (4.8), simplifies to:

$$\begin{aligned} e_t(\phi) &= \frac{1}{24 V_P} \mathbf{M} : (\nabla\nabla\phi)_P \\ &\approx \frac{1}{24 V_P} \mathbf{1} \cdot \mathbf{1} \cdot \nabla\nabla\phi_P + \mathbf{2} \cdot \mathbf{2} \cdot (\nabla\nabla\phi)_P + \mathbf{3} \cdot \mathbf{3} \cdot (\nabla\nabla\phi)_P \end{aligned}$$

4 22

The off-diagonal terms in $\mathbf{M}$ which would arise for warped control volumes are neglected. Eqn. (4.22) is still considered to be a reasonable estimate of the full

tensorial product, since the aspect ratio of the cell is taken into account. This principle can be extended to general polyhedral shapes by introducing an “orientation vector”, defined as the ratio of cell volume and the average cell surface in the given direction.

The evaluation of the $\mathbf{M} : (\nabla\nabla\phi)_P$ term in the error estimate can be simplified even further. If it is assumed that the control volume is not distorted in any way, the product scales with the square of the characteristic linear size of the control volume, Eqn. (4.12) and the magnitude of $(\nabla\nabla\phi)_P$. The error can therefore be estimated as:

$$\begin{aligned} e_t(\phi) &= \frac{1}{2V_P} |\mathbf{M} : (\nabla\nabla\phi)_P| \\ &\approx \frac{1}{24V_P} h^2 |(\nabla\nabla\phi)_P|. \end{aligned} \quad (4.23)$$

Simplification in Eqn. (4.23) further reduces the accuracy of the estimate. This is, however, a popular practice in the Finite Element community (Oden *et al.* [98]), particularly for elliptic problems.

Having in mind the situations of mesh alignment to flow gradients and high cell aspect ratios, Eqn. (4.22) has been selected in the present work as a good balance of accuracy and computational cost.

4.3.3 Measuring Numerical Diffusion

The current formulation of the Direct Taylor Series Error estimate measures the numerical diffusion only indirectly. Richardson extrapolation takes numerical diffusion effects only partially, through the solutions on meshes with different mesh spacing. An overview of numerical diffusion terms has been given in Section 3.6. Their common property is that the magnitude of the numerical diffusion coefficient scales with the mesh size. Numerical diffusion is therefore smaller on a finer mesh, which is visible in the error estimate.

The Direct Taylor Series Error estimate cannot measure numerical diffusion errors at all. If it is necessary to include it into the error estimation procedure, the following procedure can be used:

- Using the expressions for numerical diffusion from different sources, the numerical diffusion tensor is assembled for each face.
- The term $\nabla \cdot (\Gamma_{num} \cdot \nabla \phi)$ is then evaluated for every control volume. The influence of this term on the total error is obtained through normalisation, (see Section 4.4.1), using the characteristic time-scale T (defined later in Eqn. (4.33)):

$$e_{num} = \frac{|\int_{V_P} \nabla \cdot (\Gamma_{num} \cdot \nabla \phi) dV| T}{V_P}. \quad (4.24)$$

- e_{num} is added to the truncation error estimate, Eqn. (4.8).

Gradients of ϕ are estimated from the current solution. Numerical diffusion smears out local gradients as a consequence, e_{num} will be under-estimated even if the total numerical diffusion tensor is calculated exactly. In the uncommon case of negative numerical diffusion coefficients (explicit temporal discretisation, compressive differencing schemes), the opposite effect is expected.

4.4 Moment Error Estimate

The second approach to the problem of error estimation is based on the analysis of the differential equation that is being solved. An estimate of the error is derived from the properties of the exact solution.

Let us consider a steady-state scalar transport equation in the standard form, Eqn. (3.56):

$$\nabla \cdot (\rho \mathbf{U} \phi) - \nabla \cdot (\rho \Gamma_\phi \nabla \phi) = S_\phi(\phi).$$

Finite Volume discretisation produces a solution that satisfies this equation over each control volume in the integral form.

If Eqn. (3.56) were satisfied for each point in the computational domain (in the differential form), its solution would also satisfy all higher moment equations. At this stage, we shall limit ourselves to the second moment of ϕ :

$$m_\phi - \frac{1}{2} \phi^2. \quad (4.25)$$

The transport equation for m_ϕ can be derived from Eqs. (3.56) and (4.25) in the following form:

$$\nabla \cdot (\rho \mathbf{U} m_\phi) - \nabla \cdot (\rho \Gamma_\phi \nabla m_\phi) = S_\phi(\phi) \phi - \rho \Gamma_\phi (\nabla \phi \cdot \nabla \phi). \quad (4.26)$$

The numerical solution of Eqn. (3.56) is, however, of limited accuracy. There is no guarantee that Eqn. (4.26) will be satisfied for every control volume. This will be the case only if the numerical solution of Eqn. (3.56) corresponds to the exact solution on all cell centres and all faces of the mesh.

The imbalance in Eqn. (4.26) depends on the local difference between the approximate and exact solution – it can therefore be used to estimate the numerical error.

Following Eqn. (4.26), the local imbalance is equal to:

$$res_m(m_\phi) = \int_{\Omega_P} [\nabla \cdot (\rho \mathbf{U} m_\phi) - \nabla \cdot (\rho \Gamma_\phi \nabla m_\phi) - S_\phi(\phi) \phi + \rho \Gamma_\phi (\nabla \phi \cdot \nabla \phi)] dV. \quad (4.27)$$

The cell centre values of m_ϕ are calculated from the known values of ϕ using Eqn. (4.25). Following the discretisation practice presented in Chapter 3, convection, diffusion and source terms are assembled from the cell values of m_ϕ . In the calculation of face values from the convection term, Central Differencing is used.

Some attention should be given to the boundary condition treatment. On fixed value boundary faces, the value of ϕ is prescribed. This value can be used to calculate the corresponding face value of m_ϕ . On fixed gradient boundaries, a different approach is needed. The value of ϕ on the boundary face is calculated from the value in the adjacent cell, consistent with the prescribed boundary condition. The value of m_ϕ on the boundary face is then calculated from ϕ , using Eqn. (4.25).

The same principle can be extended to a general steady-state vector (tensor) transport equation. Let us consider a steady-state transport equation for a general vector property $\mathbf{a}$:

$$\nabla \cdot (\rho \mathbf{U} \mathbf{a}) - \nabla \cdot (\rho \Gamma_a \nabla \mathbf{a}) = \mathbf{S} \mathbf{u} + S_p \mathbf{a}. \quad (4.28)$$

The steady state form of the transport equation for the second moment of $\mathbf{a}$:

$$m_a = \frac{1}{2} \mathbf{a} \cdot \mathbf{a}, \quad (4.29)$$

has the following form:

$$\nabla \cdot (\rho \mathbf{U} m_a) - \nabla \cdot (\rho \Gamma_a \nabla m_a) = \mathbf{S} \mathbf{u} \cdot \mathbf{a} + 2 S p m_a - \rho \Gamma_a (\nabla \mathbf{a} : \nabla \mathbf{a}). \quad (4.30)$$

The imbalance in m_a is assembled similar to Eqn. (4.27):

$$\begin{aligned} res_m(m_a) = \int_{\Omega_P} [\nabla \cdot (\rho \mathbf{U} m_a) - \nabla \cdot (\rho \Gamma_a \nabla m_a) - \mathbf{S} \mathbf{u} \cdot \mathbf{a} \\ - 2 S p m_a + \rho \Gamma_a (\nabla \mathbf{a} : \nabla \mathbf{a})] dV. \end{aligned} \quad (4.31)$$

Unlike the Taylor series error estimation methods, the Moment Error estimate produces a scalar error for both scalar and vector properties. In the case of a vector transport equation, the resulting error estimate corresponds to the error in the magnitude of $\mathbf{a}$. The same is valid for tensor transport equations.

The Moment Error estimate measures the error through its influence on the solution. The proposed approach can be extended to higher moments of ϕ . In the case of fourth order discretisation of Eqn. (3.56), the second moment transport equation will be satisfied automatically. Effects of the numerical error can still be detected as the imbalance in transport equations for higher moments of ϕ .

The next step in the construction of the Moment Error estimate is to provide a way of estimating the magnitude of the error from the imbalance.

4.4.1 Normalisation of the Moment Error Estimate

Dimensions of $res_m(m_\phi)$ from Eqn. (4.27) are $[\phi]^2 [L]^3 / [T]$, i.e. it does not correspond to the absolute error in ϕ . In order to obtain the error magnitude, a suitable normalisation practice is required.

Eqn. 4.27 shows that $res_m(m_\phi)$ represents the volume integrated error in the transport of m_ϕ . The purpose of normalisation is to establish the influence of the imbalance to the local value of ϕ . Normalisation of $res_m(m_\phi)$ will be based on

the local transport conditions for ϕ . The effective transport consists of two parts: convective velocity and the effective diffusive transport. It is estimated as:

$$U_{trans} = |\mathbf{U}| + \frac{\Gamma_\phi}{h}. \quad (4.32)$$

The local characteristic length-scale h represents the mesh spacing. It is calculated as the ratio of cell volume and surface area, Eqn. (4.12). It should be noted that this length-scale does not have any physical meaning – it merely represents the local mesh size.

The characteristic time-scale T required for normalisation of $res_m(m_\phi)$ is calculated from the characteristic transport velocity and the local cell size:

$$T = \frac{h}{U_{trans}} = \frac{h^2}{|\mathbf{U}| h + \Gamma_\phi}. \quad (4.33)$$

Combination of Eqs. (4.33) and (4.27) enables us to estimate the solution error. The error estimate constructed this way has the dimensions of ϕ , giving an insight into the magnitude of the local error. The final form of the Moment Error estimate for a steady-state transport equation is:

$$e_m(\phi) = 2\sqrt{\frac{res_m(m_\phi) T}{V_P}}. \quad (4.34)$$

4.4.2 Consistency of the Moment Error Estimate

As noted earlier, the task of error estimation is to pinpoint the regions of the computational domain in which the difference between the numerical and real solution is large. A good error estimator should also pinpoint regions in which the solution is approximated well. If the polynomial order of local variation of the solution is lower than or equal to the order of discretisation, the error estimate should return zero.

Let us assume that the variation of ϕ over a certain region is linear. If the terms of Eqn. (3.56) are discretised to second order, the resulting ϕ field is exact. The error estimate, Eqn. (4.34), should return zero. According to Eqn. (4.25), the local variation of m_ϕ will be quadratic. If the terms of Eqn. (4.27) are also discretised to

second order accuracy, resulting $res_m(m_\phi)$ will be non-zero simply because of the discretisation error in Eqn. (4.27) – we are trying to describe a quadratic variation with a linear function. It follows that, for consistency with Eqn. (3.56), evaluation of the terms in Eqn. (4.27) should be fourth-order accurate.

The error in the evaluation of volume integrals scales with the h^4 . It can therefore be expected that the Moment Error estimate tends to the exact error faster than with h^2 , producing exact asymptotic behaviour on very fine meshes. If the mesh is too coarse, the scaling of the error estimate may be incorrect, depending on the ratio of fourth-order terms and the actual solution error.

In practice, the problem is not so serious. It rarely happens that the order of discretisation is high enough to follow the variation of the solution over the control volume exactly. If this is the case, the fourth-order terms of Eqn. (4.27) can be approximated accurately and added to the error estimate. Considering the computational overhead from the evaluation of control volume moment tensors and the limited gain in accuracy, the terms of Eqn. (4.27) will be discretised to second-order.

The influence of fourth-order terms can be partially taken into account in the normalisation of the imbalance in m_ϕ . An improved estimate of the error can be obtained if the local transport in Eqn. (4.32) is over-estimated, thus compensating for the over-estimation of the imbalance. This has been done by including the whole cell surface area into the convective transport estimate in Eqn. (4.32).

The proposed form of the Moment Error estimate measures the numerical diffusion through its influence on the solution. Coming back to the derivation of the transport equation for m_ϕ , Eqn. (4.26), it can be seen that every term in this equation has its equivalent in Eqn. (3.56). This is the reason why the exact solution of Eqn. (3.56) also satisfies Eqn. (4.26). The numerical errors introduced in the discretisation change the structure of Eqn. (3.56) on the level of the discretisation error, by introducing a numerical diffusion term. Numerical diffusion acts as an additional term in the original transport equation which does not have its equivalent in the transport equation for the moment and therefore causes an imbalance in the

volume integral for m_ϕ .

The Moment Error estimate is cheap to compute. Its behaviour on coarse meshes should be significantly better than any of the Taylor series error estimates. The influence of discretisation error is measured without the need for explicit evaluation of numerical diffusion terms. The main disadvantage of the method is its relative inaccuracy on very fine meshes.

4.5 Residual Error Estimate

The idea for the Residual Error estimate comes from similar work in Finite Element error estimation. In order to introduce the concept of residual, it is first necessary to analyse the properties of a Finite Volume solution. An appropriate normalisation of the residual is then used to estimate the magnitude of the error.

In the Finite Element method, equations are solved using the Galerkin principle (Zienkiewicz [156]) – minimisation of the weighted residual over the computational domain. The governing equations are therefore not necessarily satisfied over each finite element¹. Once the solution is calculated, the residual distribution is available. The **residual** is a function that measures how well the local solution satisfies the original governing equations. It is therefore natural to associate the level of residual with the local solution error.

The Finite Volume discretisation is derived from the integral form of the governing equation over the control volume. The numerical solution consists of cell centre and cell face values that satisfy the original equation over each cell. Cell face values are determined using an interpolation practice consistent with discretisation. The idea of the residual as a measure of accuracy is therefore not immediately obvious.

In order to introduce the residual, the basis of the Finite Volume method will be examined. For that purpose, let us assume that a numerical solution of a differential equation on a given grid with a p th order accurate Finite Volume discretisation is

¹More precisely, they are satisfied only if the the shape function can exactly follow the variation of the solution over the finite element

available: p th order accuracy implies that the function used to evaluate the surface and volume integrals consists of the first p terms of Taylor series expansion, Eqn. (4.3), and that p th order accurate face interpolation is used. The variation of the numerical solution over the control volume is therefore described by Eqn. (4.3). Let us also assume that the exact solution of the problem is available.

If the exact solution varies over the control volume faster than the p th order, the governing equation cannot be satisfied. In this situation, a control volume imbalance should exist. This is in contradiction with the basic principle of Finite Volume discretisation. In order to simplify further discussion, we shall limit ourselves to second-order accurate Finite Volume method.

The inter-point coupling in the system of algebraic equations obtained by the Finite Volume discretisation comes from the face interpolation. The cell value of ϕ depends on the values in neighbouring cells through the calculation of face values and face gradients of ϕ . Inconsistency between volume integration and face interpolation can be explained on a simple 1-D situation, Fig. 4.2.

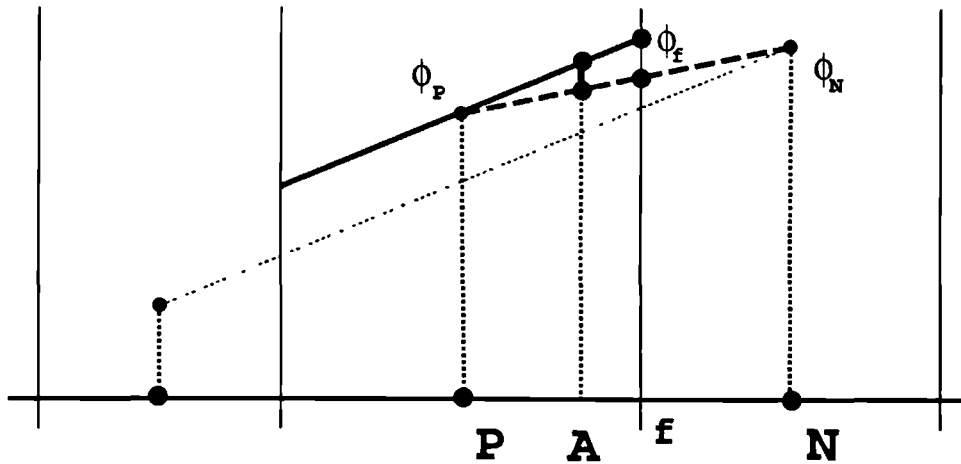


Figure 4.2: Inconsistency between face interpolation and the integration over the cell.

For volume integrals, a second-order accurate discretisation assumes linear variation of ϕ over the control volume P , Eqn. 3.3 (continuous line in Fig. 4.2).

$$\phi(\mathbf{x}) = \phi_P + (\mathbf{x} - \mathbf{x}_P) \cdot \nabla \phi_P.$$

Here, $(\nabla\phi)_P$ is calculated using discretised Gauss' theorem, Eqn. (3.26):

$$(\nabla\phi)_P = \frac{1}{V_P} \sum_f \mathbf{S}_f \phi_f.$$

On the other hand, the face value of ϕ is obtained by linear interpolation between the points P and N . The interpolated face values together with the cell-centred volume integrals satisfy the integral form of the governing equation.

Let us consider an arbitrary point A inside the control volume P . The value of ϕ in this point can be calculated in two ways:

- Following the assumed distribution of ϕ over the control volume (continuous line in Fig. 4.2), $\phi(A)$ is:

$$\phi(A) = \phi_P + (\mathbf{x}_A - \mathbf{x}_P) \cdot \nabla\phi_P, \quad (4.35)$$

since A is inside the control volume P .

- On the other hand, the face value ϕ_f for the face between P and N is obtained by interpolation. The same approach can be used for the point A which is also between P and N (dashed line in Fig. 4.2):

$$\phi(A) = f_a \phi_P + (1 - f_a) \phi_N, \quad (4.36)$$

where

$$f_a = \frac{\overline{AN}}{\overline{PN}}. \quad (4.37)$$

Values of $\phi(A)$ obtained from Eqs. (4.35) and (4.36) are identical only in the case of linear variation of the exact solution over the control volume and its neighbours, in which case the numerical error is equal to zero. If this is not the case, for every point inside the control volume two equally valid values of ϕ can be given.

On the face f , situation is even more complicated: ϕ_f obtained by interpolation, Eqn. (3.19):

$$\phi_f = f_x \phi_P + (1 - f_x) \phi_N,$$

satisfies the governing equation in the integral form. For the point just left of the face (in the P control volume), the value of ϕ is:

$$\phi_f = \phi_P + (\mathbf{x}_f - \mathbf{x}_P) \cdot \nabla \phi_P \quad (4.38)$$

and for the point just right of f , it is:

$$\phi_f = \phi_N + (\mathbf{x}_f - \mathbf{x}_N) \cdot \nabla \phi_N. \quad (4.39)$$

Let us now consider the construction of the cell residual. According to the definition used in finite elements, residual is a consequence of insufficient order of accuracy of the prescribed variation of ϕ over the control volume. It has been shown that the face values satisfying the governing equation are not consistent with the variation of ϕ used for volume integrals. A consistent face value of ϕ for the P control volume should be calculated using Eqn. (4.38). If this value is different from the interpolated ϕ_f , the integral form of the governing equation for P is not satisfied any more. The imbalance obtained this way is equivalent to the finite element definition of the residual.

Information about the error can be obtained in two ways:

- Since every face is shared by two control volumes, three different face values can be determined, Eqs. (3.19), (4.38) and (4.39). In case of the exact solution, all three values will be identical. If a face jump exists, it can be used to measure the error.
- The face values are calculated consistent with the prescribed variation of ϕ over the control volume. A residual is assembled for each control volume and normalised in an appropriate way.

Although the first approach produces the estimate of the error on the face with same dimensions as ϕ , it does not take into account the convection-diffusion balance of the equation, as the face jump in $(\nabla \phi)_f$ should also be taken into account.

The second approach produces a cell based residual, taking into account the convection-diffusion balance and the influence of mesh quality. A normalisation

practice is necessary to extract the information about the magnitude of the error. This approach is preferred because it directly incorporates all discretisation and mesh-induced errors.

For a steady-state scalar transport equation, Eqn. (3.56), the cell residual is calculated as:

$$\begin{aligned} res_P(\phi) &= \int_{V_P} [\nabla \cdot (\rho \mathbf{U} \phi) - \nabla \cdot (\rho \Gamma_\phi \nabla \phi) - Su - Sp \phi_P] dV \\ &= \sum_f \mathbf{S} \cdot [(\rho \mathbf{U})_f \phi_f - (\rho \Gamma_\phi)_f (\nabla \phi)_f] - Su V_P - Sp \phi_P V_P. \end{aligned} \quad (4.40)$$

The face values of ϕ and $\nabla \phi$ are determined from the prescribed variation of ϕ over the control volume, Eqn. (3.3):

$$\phi_f = \phi_P + (\mathbf{x}_f - \mathbf{x}_P) \cdot \nabla \phi_P \quad (4.41)$$

and

$$(\nabla \phi)_f = \nabla \phi_P. \quad (4.42)$$

Calculation of the residual does not require any special boundary treatment. Irrespective of the boundary condition, inconsistency between ϕ_f obtained using Eqn. (4.41) and the boundary face may exist. The face jump on the boundary face represents a violation of the prescribed boundary condition by the local solution, which is built into the residual.

The cell residual $res_P(\phi)$ from Eqn. (4.40) has several interesting properties:

- The residual is constructed using solely the local information about the solution and the local mesh geometry. It only requires the cell gradient of ϕ and $\overline{P}\mathbf{f}$ vectors for cell faces (see Fig. 3.15). It can be assembled on a cell-by-cell basis and is therefore cheap to compute.
- The geometry of the control volume is taken into account through the $\overline{P}\mathbf{f}$ vectors and its treatment is consistent with the discretisation practice. Mesh-induced errors are built into the calculation of the residual: the point f is

located in the middle of the face (Fig. 3.15), accounting for the skewness error, and $(\nabla\phi)_f$ does not include any non-orthogonal correction (the second term in Eqn. (3.34)) from the larger computational molecule².

- Numerical diffusion can also be detected by the residual. It has been shown that the face interpolation used in the convection term is inconsistent with the prescribed variation of ϕ over the cell irrespective of the choice of the differencing scheme. On the other hand, a comparison of Upwind and Central Differencing shows that UD produces a larger difference between ϕ_f from Eqs. (4.38) and (3.19) than CD, resulting in a larger residual.
- Residual Error estimation is directly extendible to Finite Volume discretisation of any order of accuracy. The only change in the calculation of the residual will be in the assumed variation of ϕ over the control volume.
- According to the form of the residual, the accuracy of the error estimate should not significantly change with mesh size – it should work well both on fine and coarse meshes.
- The correct scaling and asymptotic behaviour of the Residual Error estimate can be explained on a one-dimensional situation, using Fig. 4.3, as explained below.

For every control volume, two face values and a cell centre value are available. Finite Volume discretisation guarantees that this combination of face and cell centre values satisfies the governing equation in its integral form. It is therefore possible to approximate the variation of the exact solution over the cell by a parabola passing through the three available points. The error estimate is assembled using the prescribed (linear) variation of ϕ over the cell. The difference between the two is represented by the shaded region in Fig. 4.3. According to Fig. 4.3, the distribution of the error over the control volume is quadratic. This implies that the error tends to the exact error with the

²For the details of the mesh-induced errors the reader is referred to Section 3.6.3.

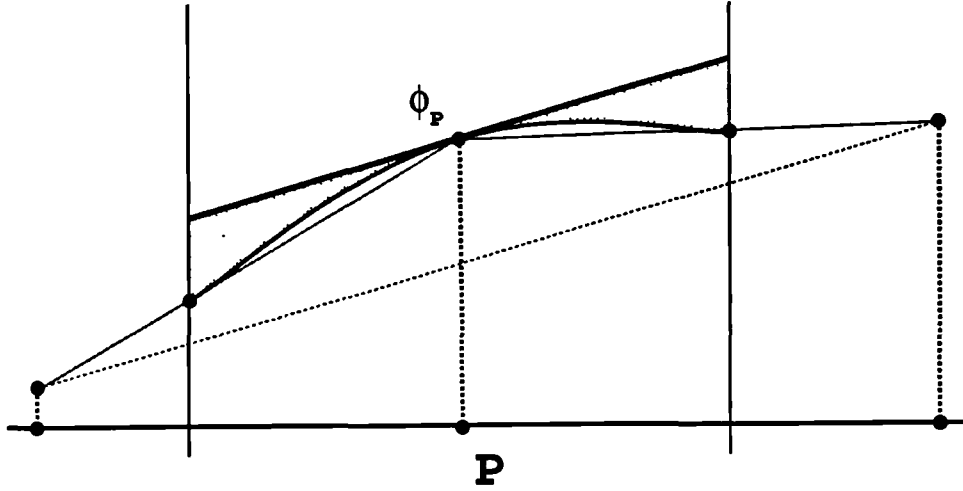


Figure 4.3: Scaling properties of the residual error estimate.

fourth power of mesh size, resulting in the asymptotically exact behaviour of the estimate. Extension of this analysis to multi-dimensional situations is straightforward.

The above properties make the Residual Error estimate an attractive error estimation tool. It is, however, still necessary to extract the information about absolute error level from the residual.

4.5.1 Normalisation of the Residual Error Estimate

Evidently, $res_P(\phi)$ has the dimensions of $[\phi][L]^3/[T]$. As in the case of the Moment Error estimate, an appropriate normalisation practice is needed to extract the error magnitude. The form of the residual, Eqn. (4.40), shows that the terms contributing to the error are the ones requiring the face values of ϕ and $\nabla\phi$, namely the convection and diffusion terms. The normalisation of $res_P(\phi)$ will therefore be based on the characteristic convection and diffusion transport for the control volume. In order to maximise the accuracy of the method, geometric information will be used on a face-by-face basis.

Consider a two-dimensional flow situation for the control volume around a point P , Fig. 4.4. Estimation of the total diffusion transport flux is simple: for each of the

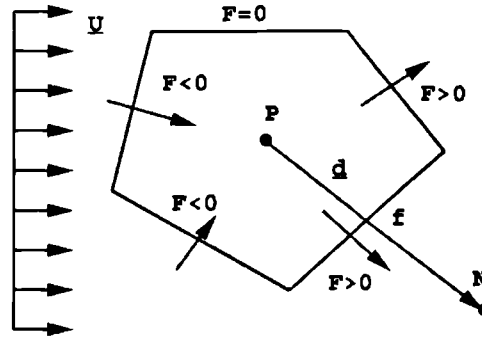


Figure 4.4: Estimating the convection and diffusion transport.

neighbours, a face diffusion coefficient is known (same as in the discretised form of the transport equation). d is used as the characteristic length and the area active in the transport is equal to S . A volume-weighted face sum of estimated diffusion transport fluxes gives the approximation of the total diffusion transport coefficient:

$$F_{diff} = \frac{1}{V_P} \sum_f \left[S_f \frac{(\rho \Gamma \phi)_f}{|d|} \right]. \quad (4.43)$$

The convection transport term requires more attention. Let us assume a locally uniform velocity field which is divergence-free. In that case, only half of the cell surface is active in the transport of interest, which reduces the influence of the residual to the local error. The convection transport coefficient can therefore be estimated as the volume weighted flux going in (or out) of the control volume:

$$F_{conv} = \frac{1}{V_P} \sum_f \max(F, 0). \quad (4.44)$$

If the velocity field is not divergence-free, discretisation of the convection term splits into two parts, with the term containing $\nabla \cdot (\rho \mathbf{U})$ being added to S_p . This will be taken into account in the normalisation, by adding this part of S_p to the normalisation factor.

Let us finally consider the influence of source term errors. No discretisation error is associated with the constant source term component – Su is considered to be exact. The error contribution from the linear part of the source, $S_p \phi_P$, scales with the error in ϕ_P and the whole of S_p should therefore be included into normalisation.

The proposed normalisation factor is therefore calculated as:

$$F_{norm} = F_{conv} + F_{diff} + Sp. \quad (4.45)$$

The final form of the Residual Error estimate is:

$$e_r(\phi) = \frac{res_P(\phi)}{V_P F_{norm}}. \quad (4.46)$$

It is interesting to notice that F_{norm} from Eqn. (4.45) actually corresponds to the volume-weighted central coefficient from the discretised transport equation for ϕ , Eqn. (3.2), where Upwind Differencing is used on the convection term.

The Residual Error estimate, Eqn. (4.46), has got many desirable properties. It is cheap to compute and does not require any tensor manipulations. Its accuracy relative to the solution accuracy is expected to be independent of the mesh size. All discretisation errors are taken into account. Overall, the accuracy of the Residual Error estimate is expected to be very good. Extension of the method to vector and tensor transport equations is straightforward. The residual and the error for a vector transport equation is a vector, where each component of the error corresponds to the error in the appropriate component in the original field. If a scalar error is preferred, it can be calculated as the magnitude of the vector error.

4.6 Local Problem Error Estimate

The Local Problem Error estimate is based on the a method of error estimation in the Finite Element field (Ainsworth [4]). This method extends and combines several existing error estimation techniques to the case of non-uniform $h - p$ Finite Element approximation on irregular meshes. It derives a strict upper bound on the solution error expressed through an error norm. Error estimation procedure requires a solution of a local residual problem. As a first step, it is necessary to calculate the residual for each control volume, as described in the previous Section.

In what follows, the error estimate is first developed for an elliptic model problem in Section 4.6.1. Section 4.6.1.1 presents the error flux balancing method for the

Finite Volume discretisation. Generalisations to the scalar transport equation and the Navier-Stokes problem are given in Sections 4.6.2 and 4.6.3, respectively. A simplified solution procedure for the local problem is given in Section 4.6.4.

For the details of the derivation of the Local Problem Error estimate, the reader is referred to [4]. This Section will only present the basic principle without the mathematical proof. Although inconsistent with the rest of this study, the notation of Ainsworth and Oden [4] will be used.

4.6.1 Elliptic Model Problem

The model problem that is being solved is:

$$L(u) = f \text{ in } \Omega, \quad (4.47)$$

with the following boundary conditions:

$$\begin{cases} u = 0 & \text{on } \Gamma_D, \\ a \hat{\mathbf{n}}_f \cdot \nabla u = g & \text{on } \Gamma_N, \end{cases} \quad (4.48)$$

where

$$L(u) = -\nabla \cdot (a \nabla u) + cu, \quad (4.49)$$

with arbitrary constants a and c and

$$\begin{aligned} \Gamma_D \cap \Gamma_N &= \emptyset, \\ \Gamma_D \cup \Gamma_N &= \partial\Omega. \end{aligned} \quad (4.50)$$

$\hat{\mathbf{n}}_f$ is a unit normal pointing in the direction of the face area vector $\mathbf{S}_f$:

$$\hat{\mathbf{n}}_f = \frac{\mathbf{S}_f}{|\mathbf{S}_f|} \quad (4.51)$$

Let us now introduce a new way of presenting the error which has been accepted as standard in Finite Element applications an **error norm**

The error norm for the elliptic problem, Eqn. (4.47), is defined as:

$$e_E^2 = \int_V (a \nabla e \cdot \nabla e + c e^2) dV. \quad 4.52$$

This norm is usually called the “energy norm”.

The property of this norm is the scaling (Ainsworth and Oden [4]):

$$e_{h,E} = \|u - u_h\|_E \leq C h^k, \quad (4.53)$$

where h is the linear size of the control volume, k is the order of approximation and C is a constant independent of h and k . Unlike other proposed ways of measuring the error, Eqn. (4.52) does not have the dimension of the solution and it cannot be directly associated with the actual error magnitude. The error norm, however, scales correctly on a non-uniform $h-p$ mesh, which is important in the case of local refinement.

Through a series of theorems, Ainsworth and Oden [4] derive the following property:

Theorem 1. *For every control volume, a local problem:*

$$-\nabla \cdot \nabla \psi_P = r_P \text{ in } \Omega_P, \quad (4.54)$$

with boundary conditions:

$$\begin{cases} \hat{\mathbf{n}}_f \cdot \nabla \psi_P = R_P & \text{on } \partial\Omega_P \setminus \Gamma_D, \\ \psi_P = 0 & \text{on } \partial\Omega_P \cap \Gamma_D, \end{cases} \quad (4.55)$$

can be used to produce a local error estimate:

$$\epsilon_P^2(\nabla \psi_P) = \int_{\Omega_K} \frac{1}{a} \nabla \psi_P \cdot \nabla \psi_P dV. \quad (4.56)$$

This error estimate provides the strict upper bound on the exact error in the energy norm:

$$e_E^2 \leq \sum_{P=1}^N \epsilon_P^2(\nabla \psi_P), \quad (4.57)$$

where

- Ω_P is the local subdomain, usually a single control volume,

- r_P is the solution residual over Ω_P , defined as:

$$r_P = f - L(u), \quad (4.58)$$

- R_P is the residual error flux,
- N is the number of subdomains (control volumes).

Proof. See Ainsworth and Oden [4]. □

The solution residual r_P is equivalent to the residual defined in the previous Section:

$$\int_{V_P} r_P(u) dV = -res_P(u). \quad (4.59)$$

As can be seen from the form of Eqn. (4.54), local problems are coupled through the boundary conditions, specifically through the residual error flux R_P . The calculation of the residual error fluxes for the Finite Element method has been discussed in detail in a number of papers [1, 4]. R_P on the cell faces is dependent on the face jumps in the solution.

Following [4], R_P is defined as:

$$R_P = \begin{cases} R_P - g - a \hat{\mathbf{n}}_f \cdot \nabla u_h & \text{on } \partial\Omega_P \cap \Gamma_N, \\ R_P - -\alpha_P \llbracket a \hat{\mathbf{n}}_f \cdot \nabla u_h \rrbracket & \text{on } \partial\Omega_P \cap \Gamma_N. \end{cases} \quad (4.60)$$

The term $\llbracket a \hat{\mathbf{n}}_f \cdot \nabla u_h \rrbracket$ is defined on each internal face of the mesh as the local jump in the gradient:

$$\llbracket a \hat{\mathbf{n}}_f \cdot \nabla u_h \rrbracket = [a \hat{\mathbf{n}}_f \cdot (\nabla u_h)_f]_P - [a \hat{\mathbf{n}}_f \cdot (\nabla u_h)_f]_N, \quad (4.61)$$

where P and N are the control volumes sharing the face and $\hat{\mathbf{n}}_f$. Similarly, the face jump in the solution is defined as:

$$\llbracket u \rrbracket = [u_h]_P - [u_h]_N. \quad (4.62)$$

Here, $[u_h]_P$ and $[\nabla u_h]_P$ are the values of u_h and ∇u_h on the face f consistent with the prescribed variation of u_h in the cell P .

We can also define an arbitrary linear interpolate of $(u_h)_f$ and $(\hat{\mathbf{n}}_f \cdot \nabla u_h)_f$:

$$\langle a \hat{\mathbf{n}}_f \cdot \nabla u_h \rangle_\alpha = \alpha_P a \hat{\mathbf{n}}_f \cdot [(\nabla u_h)_f]_P + (1 - \alpha_P) a \hat{\mathbf{n}}_f \cdot [(\nabla u_h)_f]_N \quad (4.63)$$

$$\langle u \rangle_\alpha = \alpha_P [(u_h)_f]_P + (1 - \alpha_P) [(u_h)_f]_N. \quad (4.64)$$

Nothing needs to be said about α_P in Eqs. (4.63) and (4.64) apart from that it is the same as in Eqn. (4.60).

The necessary condition for the solution of the local problem is that the error fluxes are balanced:

$$\int_{\Omega_P} r_P dV + \oint_{\partial\Omega_P} R_P dA = 0. \quad (4.65)$$

The problem of determining α_P in such a way that Eqn. (4.65) is satisfied has been discussed at length in the Finite Element community (see *eg.* Ainsworth and Oden [3]). The performance of the Local Problem Error estimate is critically dependent on this condition. If the error fluxes are not balanced, accuracy of the error estimate is degraded to such an extent that the error bound of Eqn. (4.57) becomes meaningless (see Ainsworth and Oden [4]).

In order to adopt the Local Problem Error estimate for the Finite Volume method, it is necessary to provide a procedure for calculation of balanced residual error fluxes.

4.6.1.1 Balancing Problem in Finite Volume Method

The objective of the balancing procedure is to determine the boundary conditions for local error problems, Eqn. (4.54), in such a way that the balance condition, Eqn. (4.65) is satisfied for every control volume. It will be shown that this objective can be reached without the actual calculation of α_P coefficients as a consequence of the conservative nature of the Finite Volume solution.

Error flux balancing will initially be done for the elliptic problem, Eqn. (4.47), and is subsequently generalised to the convection-diffusion and Navier-Stokes problems. In order to simplify the expressions, the subscript h will be dropped. Face values of u and ∇u are assumed when they are needed, (*i.e.* on all mesh faces).

Separating the boundary conditions, the volume residual r_P can be rewritten as:

$$\int_{\Omega_P} r_P dV = \underbrace{\int_{\Omega_P} (f - c u_P) dV}_a + \underbrace{\oint_{\partial\Omega_P \cap \Gamma_D} dS \cdot a \nabla u_P}_b. \quad (4.66)$$

Following Eqn. (4.60), the integral of the face jump can be split into two parts:

$$\oint_{\partial\Omega_P} R_P dA = \underbrace{\oint_{\partial\Omega_P \cap \Gamma_N} (g - \hat{\mathbf{n}}_f \cdot a \nabla u_P) dA}_c + \underbrace{\oint_{\partial\Omega_P \cap \Gamma_N} -\alpha [a \hat{\mathbf{n}}_f \cdot \nabla u_h] dA}_d. \quad (4.67)$$

The term (b) from Eqn. (4.66) can also be split into two, depending on the type of the prescribed boundary condition:

$$\oint_{\partial\Omega_P \cap \Gamma_D} dS \cdot a \nabla u_P = \underbrace{\oint_{\partial\Omega_P \cap \Gamma_N} dS \cdot a \nabla u_P}_e + \underbrace{\oint_{\partial\Omega_P \cap \Gamma_N \cap \Gamma_D} dS \cdot a \nabla u_P}_f. \quad (4.68)$$

The residual balance condition, Eqn. (4.65), will now be assembled part by part, using Eqs. (4.66), (4.67) and (4.68). To clarify further developments, it should be said that R_P is not defined on Γ_D , as this is not necessary (see boundary conditions for the local problem, Eqn. (4.55)).

Combining Eqn. (4.67) (c) and Eqn. (4.68) (e) yields:

$$\oint_{\partial\Omega_P \cap \Gamma_N} (g - \hat{\mathbf{n}}_f \cdot a \nabla u_P) dA + \oint_{\partial\Omega_P \cap \Gamma_N} dS \cdot a \nabla u_P = \oint_{\partial\Omega_P \cap \Gamma_N} g dA. \quad (4.69)$$

Combining Eqn. (4.67) (d) and Eqn. (4.68) (f) yields:

$$\begin{aligned} & \oint_{\partial\Omega_P \cap \Gamma_N} -\alpha [a \hat{\mathbf{n}}_f \cdot \nabla u_h] dA + \oint_{\partial\Omega_P \cap \Gamma_N \cap \Gamma_D} dS \cdot a \nabla u_P \\ &= \oint_{\partial\Omega_P \cap \Gamma_N \cap \Gamma_D} [\hat{\mathbf{n}}_f \cdot a \nabla u_P - \alpha \hat{\mathbf{n}}_f \cdot a \nabla u_P - \hat{\mathbf{n}}_f \cdot a \nabla u_N] dA \\ &= \oint_{\partial\Omega_P \cap \Gamma_N \cap \Gamma_D} [(1 - \alpha_P) a \hat{\mathbf{n}}_f \cdot \nabla u_P + \alpha_P a \hat{\mathbf{n}}_f \cdot \nabla u_N] dA \\ & - \oint_{\partial\Omega_P \cap \Gamma_N \cap \Gamma_D} a \hat{\mathbf{n}}_f \cdot \nabla u_{1-\alpha} dA. \end{aligned} \quad (4.70)$$

where see Eqn. (4.63)

$$\langle a \hat{\mathbf{n}}_f \cdot \nabla u_{1-\alpha} = (1 - \alpha_P) a \hat{\mathbf{n}}_f \cdot \nabla u_P + \alpha_P a \hat{\mathbf{n}}_f \cdot \nabla u_N. \quad (4.71)$$

Assembling the balance, Eqn. (4.65), from Eqs. (4.66), (4.69) and (4.70), it follows:

$$\int_{\Omega_P} (f - c u_P) dV + \oint_{\partial\Omega_P \cap \Gamma_N} g dA + \oint_{\partial\Omega_P \cap \Gamma_N \cup \Gamma_D} \langle a \hat{\mathbf{n}}_f \cdot \nabla u \rangle_{1-\alpha} dA = 0. \quad (4.72)$$

The result can be interpreted as follows: the interpolation method for the face fluxes $\langle a \hat{\mathbf{n}}_f \cdot \nabla u \rangle_{1-\alpha}$ should be selected in such a way that, together with the volume integral, it satisfies the original problem, Eqn. (4.47), in the integral form over each control volume.

Eqn. (4.72) provides a basis for the calculation of balanced fluxes. In the first stage, the calculation of R_P can be modified. On internal faces of the mesh, R_P is:

$$\begin{aligned} -\alpha_P [a \hat{\mathbf{n}}_f \cdot \nabla u_h] &= -\alpha_P (a \hat{\mathbf{n}}_f \cdot \nabla u_P - a \hat{\mathbf{n}}_f \cdot \nabla u_N) \\ &= (1 - \alpha_P) a \hat{\mathbf{n}}_f \cdot \nabla u_P + \alpha_P a \hat{\mathbf{n}}_f \cdot \nabla u_N - a \hat{\mathbf{n}}_f \cdot \nabla u_P \\ &= \langle a \hat{\mathbf{n}}_f \cdot \nabla u \rangle_{(1-\alpha)} - a \hat{\mathbf{n}}_f \cdot \nabla u_P. \end{aligned} \quad (4.73)$$

On boundary faces, Eqn. (4.60) can be used directly.

The remaining problem is how to determine the interpolate $\langle a \hat{\mathbf{n}}_f \cdot \nabla u \rangle_{(1-\alpha)}$. Fortunately, the Finite Volume discretisation readily provides these fluxes. As a consequence of the conservative form of the discretisation practice, the face fluxes calculated consistently with the discretisation automatically satisfy Eqn. (4.72).

For the model problem, the interpolated face gradient is

$$\langle a \hat{\mathbf{n}}_f \cdot \nabla u \rangle_{1-\alpha} = a \frac{u_N - u_P}{\mathbf{d}}. \quad (4.74)$$

In the case of non-orthogonal meshes, an explicit correction compatible with the non-orthogonality treatment should be added. The balance condition given in Eqn. (4.72) is satisfied to the solver tolerance. The calculation of R_P is straightforward, using Eqn. (4.73).

4.6.2 Generalisation to the Convection-Diffusion Problem

The extension of the Local Problem Error estimate requires the analysis of the convection term. Its contribution to the residual has been presented in Section 4.5.

The face contribution for the balanced error fluxes uses the interpolated value of ϕ to the face. For completeness, the expressions for the residual and R_P are given.

A steady-state convection-diffusion problem is specified as follows:

Find $\phi = \phi(\mathbf{x})$ such that

$$\nabla \cdot (\rho \mathbf{U} \phi) - \nabla \cdot (\rho \Gamma_\phi \nabla \phi) = Su + Sp\phi \text{ in } \Omega, \quad (4.75)$$

with boundary conditions:

$$\begin{cases} \phi = \phi_D(\mathbf{x}) & \text{on } \Gamma_D, \\ a \hat{\mathbf{n}}_f \cdot \nabla \phi = g(\mathbf{x}) & \text{on } \Gamma_N, \end{cases} \quad (4.76)$$

where

$$\begin{aligned} \Gamma_D \cap \Gamma_N &= \emptyset, \\ \Gamma_D \cup \Gamma_N &= \partial\Omega. \end{aligned}$$

The volume integral of the residual for the control volume is

$$\begin{aligned} \int_{\Omega_P} r_P dV &= \int_{\Omega_P} [Su + Sp\phi - \nabla \cdot (\rho \mathbf{U} \phi) + \nabla \cdot (\rho \Gamma_\phi \nabla \phi)] dV \\ &= Su V_P + Sp \phi_P V_P - \sum_f \mathbf{S}_f \cdot [\rho \mathbf{U}_f \phi_f - (\rho \Gamma_\phi)_f (\nabla \phi)_f], \end{aligned} \quad (4.77)$$

consistent with Eqn. 4.40).

In order to satisfy the balance condition, Eqn. (4.65), residual error fluxes are calculated as:

- On internal faces:

$$\begin{aligned} R_P &= \alpha_P F_f[\phi] - \alpha_P [\rho \Gamma_\phi \hat{\mathbf{n}}_f \cdot \nabla \phi] \\ &= F_f(-\langle \phi \rangle_{1-\alpha} + \phi_f) - (-\rho \Gamma_\phi \hat{\mathbf{n}}_f \cdot \nabla \phi_{1-\alpha} + \rho \Gamma_\phi \hat{\mathbf{n}}_f \cdot \nabla \phi_f), \end{aligned} \quad (4.78)$$

- On fixed gradient boundary faces Γ_N :

$$\begin{aligned} R_P &= \alpha_P F_f[\phi] - \alpha_P [\rho \Gamma_\phi \hat{\mathbf{n}}_f \cdot \nabla \phi] \\ &= F_f(-\phi_{1-\alpha} + \phi_f) - (-g + \rho \Gamma_\phi \hat{\mathbf{n}}_f \cdot \nabla \phi_f), \end{aligned} \quad (4.79)$$

- On fixed value boundary faces, the calculation of R_P splits into two parts:
 - For the convection part, the face jump exists:

$$\begin{aligned}(R_P)_C &= \alpha_P F_f[\phi] \\ &= F_f(-\langle \phi \rangle_{(1-\alpha)} + \phi_f),\end{aligned}\tag{4.80}$$

- For the diffusion part, Γ_D is treated as a fixed value boundary from the elliptic problem. Since the exact boundary gradient is not known, the local problem has to be solved with the fixed value boundary condition. This somewhat complicates matters, as the residual also contains the face jump from the convection term, Eqn. (4.80). An appropriate modification can be easily introduced. Before the solution of the local problem, the part of the residual from the convection term on the fixed value boundary will be removed. Now the boundary can be treated in the same way as for the elliptic problem, Section 4.6.1.

In Eqs. (4.78), (4.79) and (4.80) F_f is the face flux in respect to the direction of the face area vector (see Chapter 3):

$$F_f = \mathbf{S}_f \cdot (\rho \mathbf{U})_f,$$

where $\mathbf{S}_f$ is the face area vector. $\langle \phi \rangle_{1-\alpha}$ is calculated consistently with the convection differencing scheme used in discretisation.

The error norm for the convection-diffusion problem has the following form:

$$|e|_E^2 = |\phi - \phi_h|_E^2 = \int_V (\rho \Gamma_\phi \nabla e \cdot \nabla e + S p e^2) dV \tag{4.81}$$

The local problem is:

$$-\nabla \cdot \nabla \psi_P = r_P \text{ in } \Omega_P, \tag{4.82}$$

with boundary conditions:

$$\begin{cases} \hat{\mathbf{n}}_f \cdot \nabla \psi_P = R_P & \text{on } \partial\Omega_P \setminus \Gamma_D, \\ \hat{\mathbf{n}}_f \cdot \nabla \psi_P = (R_P)_C, \psi_P = 0 & \text{on } \partial\Omega_P \cap \Gamma_D. \end{cases} \tag{4.83}$$

The upper bound for the error in the convection-diffusion problem is the same as for the elliptic case:

$$\|e\|_E^2 \leq \sum_{P=1}^N \epsilon_P^2(\nabla\psi_P), \quad (4.84)$$

where

$$\epsilon_P^2(\nabla\psi_P) = \int_{\Omega_K} \left(\frac{1}{\rho\Gamma_\phi} \nabla\psi_P \cdot \nabla\psi_P \right) dV. \quad (4.85)$$

This concludes the extension of the Local Problem Error estimate to the convection-diffusion problem.

4.6.3 Generalisation to the Navier-Stokes Problem

The form of the Local Problem Error estimate for the Navier-Stokes equations is similar to the one for the scalar transport equation. Some points, however, need to be discussed further: the formulation of an appropriate error norm for the Navier-Stokes problem and the coupling between the pressure error and the convection-diffusion error.

Let us first formulate the incompressible steady laminar form of the Navier-Stokes problem:

Find the pair $(\mathbf{U}(\mathbf{x}), p(\mathbf{x}))$ such that

$$\begin{cases} \nabla \cdot (\rho \mathbf{U} \mathbf{U} - \nu \nabla \mathbf{U}) = \mathbf{f} - \nabla p, \\ \nabla \cdot \mathbf{U} = 0, \end{cases}$$

in Ω , with the boundary conditions:

$$\begin{cases} \mathbf{U} = \mathbf{U}_D(\mathbf{x}), \nabla p = 0 & \text{on } \Gamma_D, \\ \hat{\mathbf{n}}_f \cdot \nabla \mathbf{U} = 0, p = p_N(\mathbf{x}) & \text{on } \Gamma_N, \end{cases} \quad (4.86)$$

where:

$$\begin{aligned} \Gamma_D \cap \Gamma_N &= \emptyset, \\ \Gamma_D \cup \Gamma_N &= \partial\Omega. \end{aligned}$$

4.6.3.1 Error Norm for the Navier-Stokes System

In order to simplify the discussion, from here on all the variables will be considered to be dimensionless, with the characteristic scales equal to unity (following Girault and Raviart [52]). The uniqueness of the solution of the Navier-Stokes problem has been discussed by Oden *et al.* [101], Girault and Raviart [52] and Temam [131]. If the uniqueness condition is satisfied, the following property of the numerical solution can be proven (see Oden *et al.* [101], or Girault and Raviart [52]):

Theorem 2. *Suppose that the conditions for the uniqueness of the solution for the Navier-Stokes problem hold (see [101, 52]). Let $(\mathbf{U}, p)$ be the solution of Eqn. (4.86). Then, for ν sufficiently large, there exists a mesh spacing h_0 such that for all $h \leq h_0$, the discretised form of Eqn. (4.86) has a unique solution $(\mathbf{U}_h, p_h)$ and*

$$\lim_{h \rightarrow 0} (\|\mathbf{U} - \mathbf{U}_h\|_1 + \|p - p_h\|_0) = 0. \quad (4.87)$$

If, in addition, the solution $(\mathbf{U}, p)$ of Eqn. (4.86) is in the space of accessible functions of the order $k+1$, where k is the order of accuracy of the discretisation method, then a constant $C > 0$ exists, independent of k , such that

$$\|\mathbf{U} - \mathbf{U}_h\|_1 + \|p - p_h\|_0 \leq C h^k. \quad (4.88)$$

Proof. See [52]. □

The norms in Theorem 2 are defined as:

$$\|\mathbf{v}\|_1^2 = \int_{\Omega} \nu (\nabla \mathbf{v} : \nabla \mathbf{v}) dV, \quad (4.89)$$

$$\|q\|_0^2 = \int_{\Omega} q^2 dV. \quad (4.90)$$

Oden, [101], defines the error in the solution of Eqn. (4.86) as:

$$\mathbf{e} = \mathbf{U} - \mathbf{U}_h, \quad (4.91)$$

$$E = p - p_h \quad (4.92)$$

and introduces a new “star” norm $(\mathbf{e}, E)_*$. It is consequently proven (Oden, [101]) that the new norm is equivalent to the norm for Navier-Stokes problem:

$$(\mathbf{e}, E)_* = \|\mathbf{U} - \mathbf{U}_h\|_1 + \|p - p_h\|_0. \quad (4.93)$$

4.6.3.2 Formulation of the Local Problem

Following the derivation of the Local Problem Error estimate, it can be shown that there exists an upper error bound on the error in the “star” norm.

Theorem 3. *Let $\mathbf{m}_P$ be the solution of the local problem:*

$$-\nabla \cdot \nabla \mathbf{m}_P = \mathbf{r}_P \text{ in } \Omega_P, \quad (4.94)$$

with boundary conditions:

$$\begin{cases} \hat{\mathbf{n}}_f \cdot \nabla \mathbf{m}_P = \mathbf{R}_P & \text{on } \partial\Omega_P \setminus \Gamma_D, \\ \hat{\mathbf{n}}_f \cdot \nabla \mathbf{m}_P = (\mathbf{R}_P)_C, \mathbf{m}_P = \mathbf{0} & \text{on } \partial\Omega_P \cap \Gamma_D. \end{cases} \quad (4.95)$$

The residual $\mathbf{r}_P$ and face jumps $\mathbf{R}_P$ and $(\mathbf{R}_P)_C$ are defined as:

$$\mathbf{r}_P = \int_{\Omega_P} [\mathbf{f} - \nabla p - \nabla \cdot (\rho \mathbf{U} \mathbf{U} - \nu \nabla \mathbf{U})] dV, \quad (4.96)$$

on internal faces:

$$\mathbf{R}_P = F_f(-\langle \mathbf{U} \rangle_{(1-\alpha)} + \mathbf{U}_f) - |\mathbf{S}| [-\langle \nu \hat{\mathbf{n}}_f \cdot \nabla \mathbf{U} \rangle_{(1-\alpha)} + \nu \hat{\mathbf{n}}_f \cdot (\nabla \mathbf{U})_f], \quad (4.97)$$

on zero gradient boundary faces:

$$\mathbf{R}_P = F_f(-\langle \mathbf{U} \rangle_{(1-\alpha)} + \mathbf{U}_f) - |\mathbf{S}| \nu \hat{\mathbf{n}}_f \cdot \nabla \mathbf{U}_f \quad (4.98)$$

and on fixed value boundary faces:

$$(\mathbf{R}_P)_C = F_f(-\langle \mathbf{U} \rangle_{(1-\alpha)} + \mathbf{U}_f). \quad (4.99)$$

Then, the error $(\mathbf{e}, E)$ of the discrete solution of the Navier-Stokes problem Eqn. (4.86) satisfies the following bound:

$$\|(\mathbf{e}, E)\|_*^2 \leq \sum_{P=1}^N \epsilon_P^2 (\nabla \mathbf{m}_P, \nabla \cdot \mathbf{U}_P), \quad (4.100)$$

where N is the number of subdomains (finite volumes).

The local error estimate $\epsilon_P^2(\nabla \mathbf{m}_P, \nabla \cdot \mathbf{U}_P)$ is defined as:

$$\epsilon_P^2(\nabla \mathbf{m}_P, \nabla \cdot \mathbf{U}_P) = \int_{\Omega_P} \left(\frac{1}{\nu} \nabla \mathbf{m}_P : \nabla \mathbf{m}_P \right) dV + \int_{\Omega_P} (\nabla \cdot \mathbf{U}_P)^2 dV. \quad (4.101)$$

Proof. See [101]. □

The only difference between the convection-diffusion and the Navier-Stokes problem is the treatment of the additional source term: the pressure gradient. In the convection-diffusion problem, constant source terms are considered to be exact, whereas here the pressure gradient term carries a certain numerical error associated with the divergence of velocity.

4.6.4 Solution of the Local Problem

The results on the upper bound for the error, Eqs. (4.57), (4.84) and (4.100), assume that the local problem is solved exactly over each subdomain. This is, however, potentially expensive and for purposes of error estimation not really necessary. In this Section, a simple method for the approximate solution of the local problem is proposed in the framework of second-order accurate Finite Volume discretisation. Numerical experiments show that this method is appropriate for error estimation.

Let us first take another look at the definition of the local problem for the elliptic equation:

$$-\nabla \cdot \nabla \psi_P = r_P \text{ in } \Omega_P, \quad (4.102)$$

with boundary conditions:

$$\begin{cases} \hat{\mathbf{n}}_f \cdot \nabla \psi_P = R_P & \text{on } \partial\Omega_P \setminus \Gamma_D, \\ \psi_P = 0 & \text{on } \partial\Omega_P \cap \Gamma_D. \end{cases} \quad (4.103)$$

The boundary conditions show a certain peculiarity:

- If the control volume has a face on the fixed value boundary Γ_D , the problem is well-posed.
- If, on the other hand, there are no fixed value boundary faces on the control volume, the problem is indeterminate up to a constant.

The actual error estimate does not require ψ_P , but only the volume integral of its gradient. It can therefore be obtained without knowing the actual level of ψ_P . The solution procedure for the local problem is set up in such a way that the level of ψ_P is calculated only when the problem is well-posed.

Let us first discuss the situation when the cell does not have any fixed value boundary faces.

4.6.4.1 Solution of the Indeterminate Local Problem

The local error estimate ϵ_P is calculated as:

$$\epsilon_P^2(\nabla\psi_P) = \int_{\Omega_K} \frac{1}{a} \nabla\psi_P \cdot \nabla\psi_P dV.$$

It will be assumed that $\nabla\psi_P$ changes linearly through the cell – the cell will not be further subdivided. It is therefore only necessary to determine the value of $\nabla\psi_P$ in the cell centre. This will be done by formally solving the local residual problem on a mesh consisting of a single cell.

The gradient of ψ in the cell centre is calculated using the discretised version of the Gauss' theorem:

$$\nabla\psi_P = \frac{1}{V_P} \sum_f \mathbf{S} \psi_f \quad (4.104)$$

and

$$\psi_f = \psi_P + (\mathbf{x}_f - \mathbf{x}_P) \cdot \nabla\psi_P. \quad (4.105)$$

It is further assumed that $\nabla\psi_P$ and $\nabla\psi_f$ are identical. ψ_f is then equal to:

$$\psi_f = \psi_P + (\mathbf{x}_f - \mathbf{x}_P) \cdot \nabla\psi_f. \quad (4.106)$$

Using Eqs. (4.104) and (4.106), it follows:

$$\nabla\psi_P = \frac{1}{V_P} \sum_f \mathbf{S} [(\mathbf{x}_f - \mathbf{x}_P) \cdot \nabla\psi_f], \quad (4.107)$$

where the ψ_P part disappears.

The boundary condition prescribes $\hat{\mathbf{n}}_f \cdot \nabla \psi_P$ on each face. A further simplification is necessary to use it with Eqn. (4.107). Neglecting mesh non-orthogonality, it follows that:

$$(\mathbf{x}_f - \mathbf{x}_P) \cdot \nabla \psi_f = |\mathbf{x}_f - \mathbf{x}_P| \hat{\mathbf{n}}_f \cdot \nabla \psi_f. \quad (4.108)$$

Putting Eqs. (4.107) and (4.108) together, the following expression for $\nabla \psi_P$ is obtained:

$$\nabla \psi_P = \frac{1}{V_P} \sum_f \mathbf{S}_f |\mathbf{x}_f - \mathbf{x}_P| \hat{\mathbf{n}}_f \cdot \nabla \psi_f = \frac{1}{V_P} \sum_f \mathbf{S}_f |\mathbf{x}_f - \mathbf{x}_P| R_P. \quad (4.109)$$

4.6.4.2 Solution of the Determinate Local Problem

The solution procedure for the determinate local problem will be set up in such a way that the result from the previous Section can be used. The only information missing is the gradient on fixed value boundary faces.

In order to calculate this gradient, the local problem will again be formally solved on the mesh with a single control volume.

The local problem is discretised Eqn. (4.54) using the discretised form of the Gauss' theorem, Eqn. (4.104):

$$-\sum_f \mathbf{S}_f (\nabla \psi_P)_f = r_P V_P. \quad (4.110)$$

The faces with the fixed gradient boundary condition will be called the “g”-faces and the faces with the fixed value boundary condition the “v”-faces. The sum from Eqn. (4.110) splits into two:

$$\sum_f \mathbf{S}_f (\nabla \psi_P)_f = \sum_g \mathbf{S}_g (\nabla \psi_P)_g + \sum_v \mathbf{S}_v (\nabla \psi_P)_v. \quad (4.111)$$

For the “g”-faces, the boundary condition is given:

$$\mathbf{S}_g (\nabla \psi_P)_g = |\mathbf{S}_g| \hat{\mathbf{n}}_g \cdot (\nabla \psi_P)_g = |\mathbf{S}_g| (R_P)_g, \quad (4.112)$$

whereas for the “v”-faces we have:

$$\mathbf{S}_v (\nabla \psi_P)_v = |\mathbf{S}_v| \hat{\mathbf{n}}_v \cdot (\nabla \psi_P)_v = |\mathbf{S}_v| \frac{(\psi_P)_v - (\psi_P)_P}{d}. \quad (4.113)$$

Here, $(\psi_P)_P$ is the solution of the local problem in the cell centre. $(\psi_P)_v$ is equal to zero, following the boundary condition on the local problem, Eqn. (4.55).

$(\psi_P)_P$ is therefore:

$$(\psi_P)_P = \frac{\int_{\Omega_P} r_P dV + \sum_g \mathbf{S} (R_P)_g}{\sum \mathbf{S} \frac{1}{d}} \quad (4.114)$$

and the face gradients for the “v”-faces are

$$\hat{\mathbf{n}}_f \cdot (\nabla \psi_P)_v = \frac{-(\psi_P)_P}{|d|}. \quad (4.115)$$

All face gradients are now known and Eqn. (4.109) can again be used.

This concludes the solution procedure for the local problem in the elliptic case. Generalisation to other types of the local problems is straightforward.

4.6.5 Application of the Local Problem Error Estimate

The Local Problem Error estimate does not describe the magnitude of the solution error directly. It is therefore necessary to establish what kind of information about the error can be recovered from this error norm.

The optimal mesh, irrespective of its size, is the one in which the error is uniformly distributed through the whole domain. The actual level of the error depends on the problem that is being solved and the number of control volumes. It follows that in an optimal mesh the error gradients are equal to zero. The error norm from Eqn. (4.52) is based on the magnitude of the error gradient and therefore provides information about the mesh quality.

The upper bound on the error norm, Eqn. (4.57), describes the relation between the solution error and the solution of the local problem. In most cases, error flux boundary conditions cause the indeterminacy of the local problem up to a constant (see Section 4.6.4). This indeterminacy implies that no information about the magnitude of the error can be recovered from the Local Problem Error estimate.

The energy (dissipation) norm for the incompressible Navier-Stokes equations can, however, be compared with the total dissipation in the system. Oden [102] describes an error-controlled adaptive refinement procedure based on the Local Prob-

lem Error estimate. The accuracy of the solution is considered satisfactory when the error is reduced to 3 – 5% of the total dissipation.

4.7 Error Estimation for Transient Calculations

Error estimation in transient calculations can be seen as an extension of steady-state error estimation. Apart from the error coming from spatial resolution, additional temporal effects need to be taken into account.

In transient problems, three separate accuracy questions can be posed:

- In the first instance, it is necessary to measure the error introduced by each time-step of the transient calculation. Both the spatial and temporal error should be taken into account.
- The second question is concerned with the accuracy of the solution after a certain number of time-steps. To answer this question, it is necessary to follow the evolution of the error in time.
- Finally, in transient cases with adaptive refinement and tracking of flow features, results of error estimation are used to change the mesh and the time-step size. It is therefore necessary to estimate the spatial and temporal errors separately. Decision on mesh refinement will be based on the spatial error only. The ratio of spatial and temporal error can also be used for automatic time-step control – the requirement for the equivalent temporal and spatial accuracy enables us to estimate the optimal time-step size.

We shall start the analysis of the transient error with the formulation of the temporal residual. Section 4.7.2 presents a procedure for separate estimation of spatial and temporal errors. Finally, Section 4.7.3 gives an insight into the transportive properties of the error.

4.7.1 Residual in Transient Calculations

Let us consider the integral form of the transport equation for ϕ , Eqn. (3.8):

$$\begin{aligned} \int_t^{t+\Delta t} \left[\frac{\partial}{\partial t} \int_{V_P} \rho \phi dV + \int_{V_P} \nabla \cdot (\rho \mathbf{U} \phi) dV - \int_{V_P} \nabla \cdot (\rho \Gamma_\phi \nabla \phi) dV \right] dt \\ = \int_t^{t+\Delta t} \left(\int_{V_P} S_\phi(\phi) dV \right) dt. \end{aligned}$$

It has been shown that the assumption of linear variation in space and time leads to the Crank-Nicholson form of discretisation, Eqn. (3.41):

$$\begin{aligned} \frac{\rho_P \phi_P^n - \rho_P \phi_P^o}{\Delta t} V_P + \frac{1}{2} \sum_f F \phi_f^n - \frac{1}{2} \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f^n \\ + \frac{1}{2} \sum_f F \phi_f^o - \frac{1}{2} \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot (\nabla \phi)_f^o \\ = Su V_P + \frac{1}{2} Sp V_P \phi_P^n + \frac{1}{2} Sp V_P \phi_P^o. \end{aligned}$$

Section 4.5 gives the residual error analysis for a steady-state situation. The convection-diffusion residual for a steady-state problem was assembled from the prescribed linear variation of ϕ in space. This spatial residual has the following form Eqn. (4.40) :

$$\begin{aligned} res_P(\phi) &= \int_{V_P} [\nabla \cdot (\rho \mathbf{U} \phi) - \nabla \cdot (\rho \Gamma_\phi \nabla \phi) - Su - Sp \phi_P] dV \\ &= \sum_f \mathbf{S} \cdot [(\rho \mathbf{U})_f \phi_f - (\rho \Gamma_\phi)_f (\nabla \phi)_f] - Su V_P - Sp \phi_P V_P. \end{aligned}$$

Eqn. (4.40) estimates the error in transport terms of Eqn. (3.41). The complete residual can therefore be assembled by including the transient term:

$$res_F(\phi) = \frac{\rho_P \phi_P^n - \rho_P \phi_P^o}{\Delta t} V_P + \frac{1}{2} res_P(\phi^n) + \frac{1}{2} res_P(\phi^o), \quad (4.116)$$

where $res_P(\phi^n)$ and $res_P(\phi^o)$ are calculated using Eqn. (4.40).

Eqn. 4.116 takes into account all deviations of second-order accuracy, thus measuring all discretisation errors. In order to obtain the error magnitude, the normalisation described in Section 4.5.1 is used.

4.7.2 Spatial and Temporal Error Contributions

The previous Section describes the formulation of the residual for transient calculations, including both the spatial and temporal error terms. It would be useful to know how much of the total error is caused by spatial and how much by temporal discretisation.

In transient tracking calculations, changes in the computational mesh follow certain features that require high spatial resolution (moving shocks, pressure waves, flame fronts *etc.*). The tracking effect is created by a region of fine mesh that travels through the domain with the feature of interest. Since the speed and the direction of its propagation are not known in advance, the adaptive algorithm relies on the error estimate to add and remove computational points. The tracking problem can be described as an adaptive calculation with simultaneous refinement and unrefinement. For such calculations, it is advisable to neglect the temporal part of the error, as it obscures the spatial error sources of interest. From the refinement point of view, the transient term is therefore considered to be exact. The spatial error alone gives more accurate information about the location of the tracked flow feature.

For first-order temporal schemes (Euler Implicit and Explicit discretisation), the spatial error residual is equal to:

$$res_L^{EI,EX}(\phi) = \frac{\rho_P \phi_P^n - \rho_P \phi_P^o}{\Delta t} V_P + res_P(\phi^n). \quad (4.117)$$

Similarly, for Backward Differencing in time:

$$res_L^{BD}(\phi) = \frac{\frac{3}{2}\phi^n - 2\phi^o + \frac{1}{2}\phi^{oo}}{\Delta t} V_P + res_P(\phi^n). \quad (4.118)$$

For Crank-Nicholson discretisation, the spatial error estimate is the same as for the first-order temporal schemes, as the rate of change of the spatial error in time is neglected. For other methods of temporal discretisation, the temporal error consists of two parts: the temporal error of the Crank-Nicholson method and the additional temporal numerical diffusion error (see Section 3.6.2). The error estimate can be derived from the difference between Eqs. (4.116) and (4.117) or Eqs. (4.116) and (4.118), respectively.

For the first-order temporal schemes, the temporal component of the residual is equal to:

$$\begin{aligned} res_T^{EI,EX}(\phi) &= res_F(\phi) - res_L^{EI,EX}(\phi) \\ &= -\frac{1}{2} res_P(\phi^n) + \frac{1}{2} res_P(\phi^o). \end{aligned} \quad (4.119)$$

An insight into Eqn. (4.119) can be given if it is assumed that spatial discretisation produces no error. In that case $res_L^{BD}(\phi) = 0$, and from Eqn. (4.117) it follows:

$$res_P(\phi^n) = -\frac{\rho_P \phi_P^n - \rho_P \phi_P^o}{\Delta t} V_P. \quad (4.120)$$

Using Eqn. (4.120) for ϕ^n and ϕ^o in Eqn. (4.119), it follows:

$$\begin{aligned} res_T^{EI,EX}(\phi) &= \frac{1}{2} \frac{\rho_P \phi_P^n - \rho_P \phi_P^o}{\Delta t} V_P - \frac{1}{2} \frac{\rho_P \phi_P^o - \rho_P \phi_P^{oo}}{\Delta t} V_P \\ &= \frac{1}{2} \frac{\rho_P \phi_P^n - 2 \rho_P \phi_P^o + \rho_P \phi_P^{oo}}{\Delta t} V_P. \end{aligned} \quad (4.121)$$

which approximates the second derivative in time. This is consistent with the leading term of the truncation error for the first-order temporal schemes. The transient error estimate therefore correctly estimates the temporal error in the case of the first-order accurate temporal discretisation.

The separation of the residual into the spatial and temporal contribution enables us to look into the issue of optimal time-step size. Equivalent temporal and spatial accuracy implies that the peak error magnitudes from temporal and spatial discretisation are the same. The influence of the spatial and temporal error is measured by the time-step indicator

$$t_{ref} = \frac{\max(res_T(\phi))}{\max(res_F(\phi))}. \quad (4.122)$$

For $t_{ref} > \frac{1}{2}$, temporal error dominates. It is more efficient to reduce the time-step than to refine the mesh; for $t_{ref} < \frac{1}{2}$, the situation is opposite. The time-step indicator should, however, be used only as a global indicator of the relative quality of temporal and spatial discretisation, as it will typically change in time.

4.7.3 Evolution Equation for the Error

Let us consider a scalar transport equation in the standard form, Eqn. (3.2)

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho \mathbf{U} \phi) - \nabla \cdot (\rho \Gamma_{\phi} \nabla \phi) = S_{\phi}(\phi).$$

In transient calculation, a residual is created for every time-step as a consequence of the discretisation error. The error is built into the solution and follows its development. The accuracy of the solution after several time-steps will therefore depend not only on the current discretisation error, but on the errors in all previous time-steps – the error in transient calculations has potentially cumulative effects.

In order to estimate the total solution error, it is necessary to follow its development from the initial condition. It is assumed that the initial condition is exact, *i.e.* that no error has been introduced by its discrete representation.

The local error source is expressed as the cell residual. It includes both the effects of temporal and spatial discretisation.

Since the error is transported with the solution, both ϕ and the error are subject to the same transport process. It is therefore possible to write the **transport equation for the error** in the following form:

$$\frac{\partial \rho e}{\partial t} + \nabla \cdot (\rho \mathbf{U} e) - \nabla \cdot (\rho \Gamma_{\phi} \nabla e) = S_e. \quad (4.123)$$

Boundary conditions for Eqn. (4.123) are easily derived. On fixed value boundaries for ϕ , no numerical error is introduced – e is therefore fixed to zero. On all other types of boundary conditions, e mimics the behaviour of ϕ . The local error source, S_e shows the influence of discretisation to local accuracy. It would be natural to estimate S_e as the volume-weighted cell residual:

$$S_e = \frac{res_F(\phi)}{V_P}. \quad (4.124)$$

The solution of Eqn. (4.123) should in theory produce the “exact” distribution of the numerical error through the domain. If the equation were solved exactly with exact error sources, it would be possible to re-create the exact solution by adding this error to the numerical solution. This is, however, not reasonable – the

numerical solution of Eqn. (4.123) is subject to the same discretisation problems as the original transport equation. Additional uncertainty comes from the estimate of the error source S_e . An example one-dimensional convection study will be given in Section 4.8. Attempts to solve the equivalent of Eqn. (4.123) for steady-state problems have produced disappointing results. In the author's opinion, this is a consequence of the local equilibrium of the error source and transport, which changes the nature of the transported error.

4.8 Numerical Examples

In order to examine the accuracy of error estimation methods presented in this Chapter, several numerical examples will be considered. The selected test cases have analytical solutions, which allows comparison between the exact and estimated errors. Although the presented cases do not represent "real" flow situations, they share some properties with them and provide an useful insight into the accuracy and scaling properties of error estimates.

4.8.1 Line Source in Cross-Flow

The first test case is a convection-diffusion problem for a scalar variable *eg.* temperature or concentration. It consists of a fixed strength line source of a passive scalar in a fixed uniform velocity field. In order to avoid the singularity at the line source, it is located outside the computational domain. The analytical solution for this test case can be found in Hinze [64]:

$$\phi(x, y) = \frac{S^*}{2\pi\Gamma} K_0\left(\frac{U_1\sqrt{x^2 + y^2}}{2\Gamma}\right) e^{-\frac{U_1 x}{2\Gamma}}. \quad (4.125)$$

Here

- x and y are the spatial coordinates. The origin of the coordinate system is located at the source, with the x coordinate pointing in the direction of the velocity vector,

- $S^* = 16.67 [\phi]/s$ is the strength of the source,
- $\Gamma = 0.05 m^2/s$ is the diffusion coefficient,
- $U_1 = 1 m/s$ is magnitude of the velocity,
- K_0 is the modified Bessel function of the second kind and zero order.

In order to examine the scaling properties of the error, the problem will be solved on a series of uniformly refined meshes. The exact and estimated errors will then be plotted against the number of computational points.

Two situations of mesh-to-flow alignment will be presented: a uniform orthogonal mesh aligned with the flow in Section 4.8.1.1, and a non-uniform non-aligned non-orthogonal mesh, Section 4.8.1.2.

4.8.1.1 Mesh Aligned with the Flow

The test setup for this situation is given in Fig. 4.5. The size of the domain is

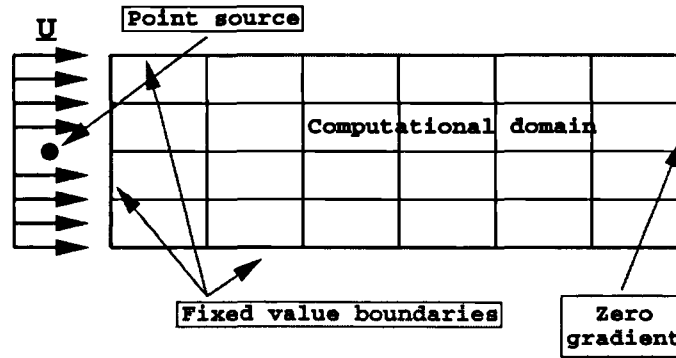


Figure 4.5: Line source in cross-flow: mesh aligned with the flow.

$4 \times 1 m$, $0.05 m$ downstream of the source. The analytical solution is prescribed on fixed value boundaries. Meshes used in this comparison range from 50 to 51520 CV-s.

Fig. 4.6 shows the exact solution on the 80×41 CV-s. The distribution of the exact and estimated errors is given in Figs. 4.7, 4.8, 4.9 and 4.10.

Comparing Figs. 4.8, 4.9 and 4.10 with Fig. 4.7, two parts of the exact error can be recognised. The region of the highest exact error corresponds to the highest

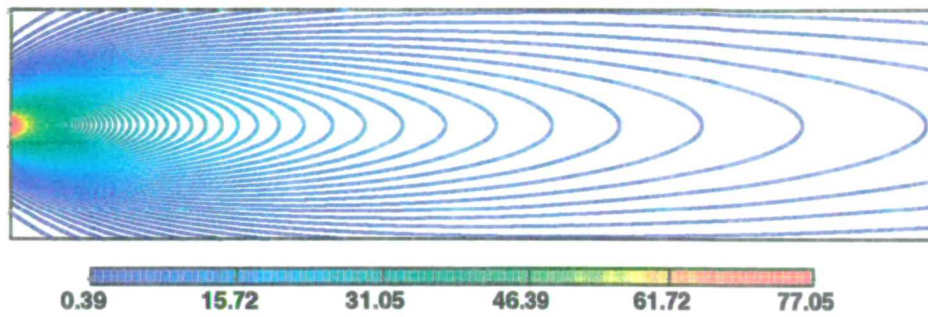


Figure 4.6: Aligned mesh: exact solution.

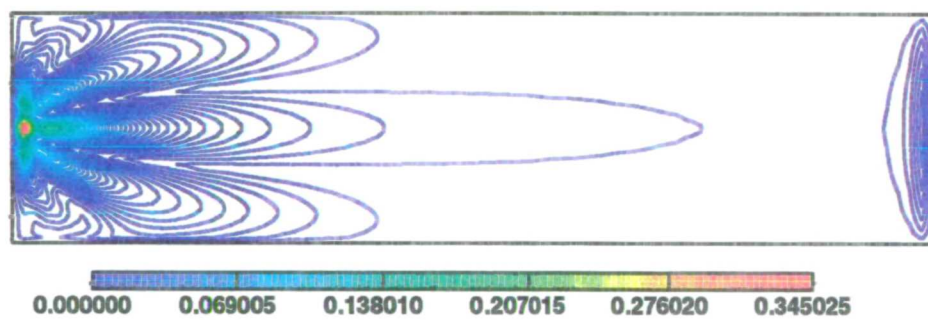


Figure 4.7: Aligned mesh: Exact error magnitude.

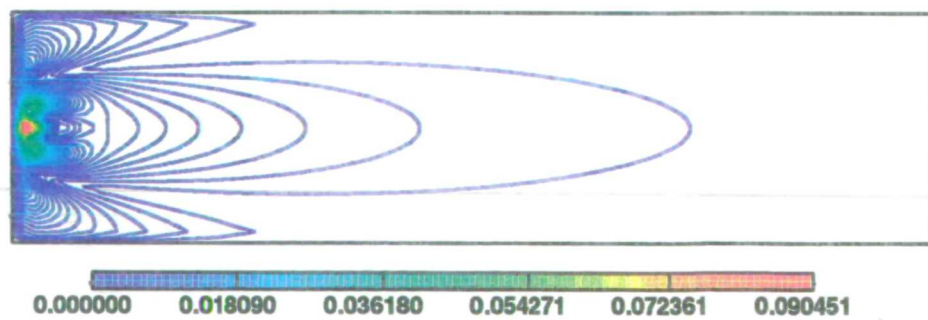


Figure 4.8: Aligned mesh: Direct Taylor Series Error estimate.

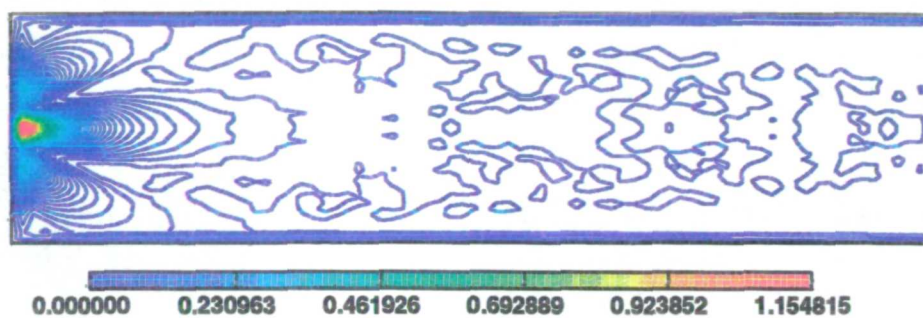


Figure 4.9: Aligned mesh: Moment Error estimate.

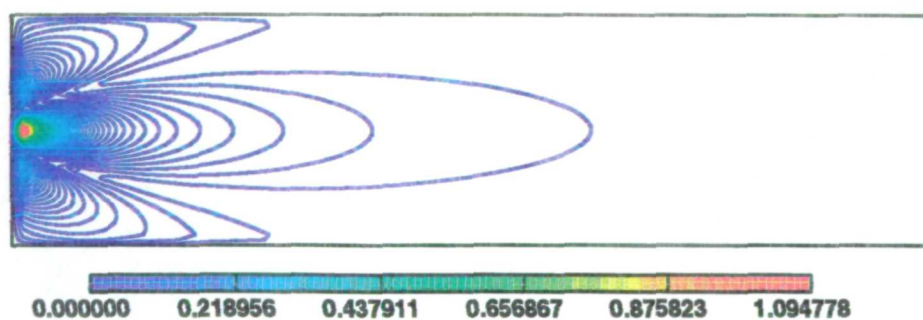


Figure 4.10: Aligned mesh: Residual Error estimate.

local discretisation error. This error source has been accurately detected by all error estimates. The secondary effect of this high error can be seen just downstream from the error source. Here, the exact error is still large, but the estimates do not pick up its source. The error is transported from the region with high discretisation error and will disappear when the source is removed. The secondary error cannot be detected by the error estimate because it does not result from the local discretisation error. As the mechanism of error transport is not fully understood, we shall assume that the error source only creates a local error. The results in the present study show that this simplification does not degrade the accuracy of error estimation and that it is particularly useful from the point of view of local mesh refinement, where the detection of error sources is more important than the actual magnitude.

Let us now consider the scaling properties of the error and error estimates. Figs. 4.11 and 4.12 show the reduction of the mean and maximum error with the number of control volumes. It is expected that the error will reduce with the square of the mesh size, following the second order accuracy of the method.

In two spatial dimensions, the number of computational points quadruples if the mesh spacing is halved in each direction. The slope of the curve in Fig. 4.11 should therefore be -1 . Slight variations of the slope and the rapid drop in the maximum error on coarse meshes are attributed to the discretisation of the exact boundary condition.

The advantage of the Moment and Residual Error estimates over the Direct Taylor Series Error estimate can be clearly seen, both in the mean and maximum error. The Moment and Residual estimates give almost identical results on coarse meshes. As the solution gets more accurate, the Moment Error estimate loses accuracy – the magnitude of the neglected fourth-order terms becomes similar to the magnitude of the solution error. The Residual Error estimate consistently gives good results.

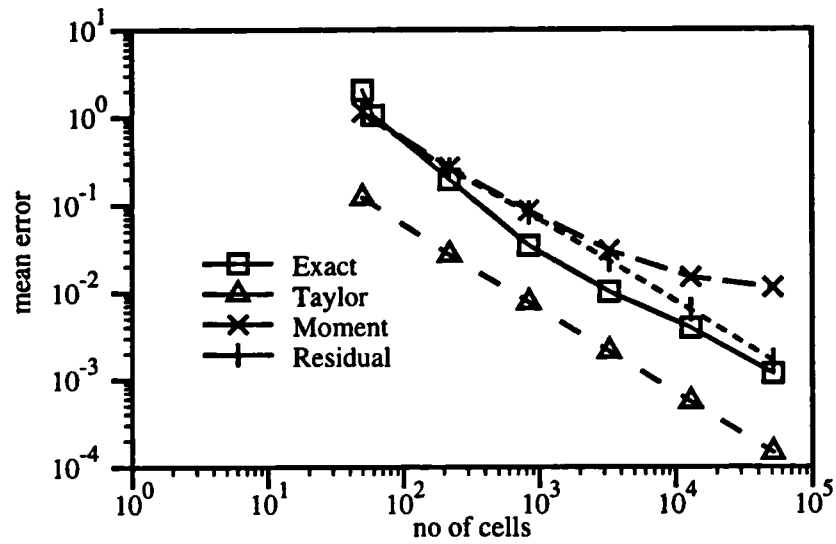


Figure 4.11: Aligned mesh: scaling of the mean error.

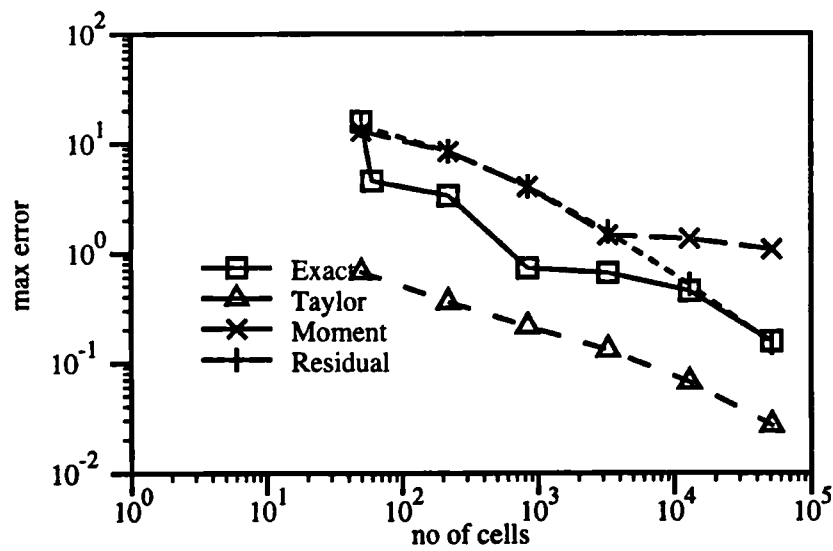


Figure 4.12: Aligned mesh: scaling of the maximum error.

4.8.1.2 Non-Orthogonal Non-Aligned Mesh

Let us now consider the effects of mesh-to-flow alignment and non-orthogonality on the accuracy of the solution. Skewness and non-orthogonality errors are introduced by the mesh. In the first instance, CD will be used for the convection term. The test setup for this situation is given in Fig. 4.13.

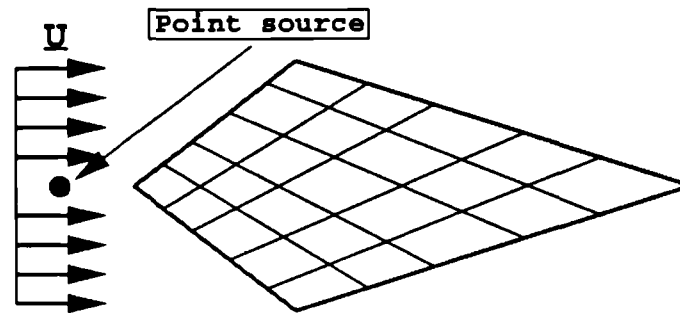


Figure 4.13: Line source in cross-flow: non-orthogonal non-aligned mesh.

The dimensions of the domain are $3 \times 1.41 \text{ m}$, with the same diffusion coefficient and relative position to the source as in the previous case. Boundary conditions from the analytical solution are prescribed on all boundaries.

The exact solution on the 40×40 mesh is shown in Fig. 4.14. The exact and estimated error distribution are given in Figs. 4.15, 4.16, 4.17 and 4.18. All error estimation methods pick up the error source well.

The scaling of the exact and estimated error with the mesh size is shown in Figs. 4.19 and 4.20. In the case of mean error, all estimates show comparable accuracy. The Direct Taylor Series estimate, however, under-estimates the error. Discretisation errors introduced by mesh distortion did not noticeably change the order of the method.

In order to introduce even more numerical diffusion, Upwind Differencing is used for the convection term. The scaling of the mean and maximum error is shown in Figs. 4.21 and 4.22. The slope of the curve is now closer to $-\frac{1}{2}$. It can also be seen that both mean and maximum error are much higher than with Central Differencing (Figs. 4.11 and 4.12), illustrating the importance of second-order convection discretisation.

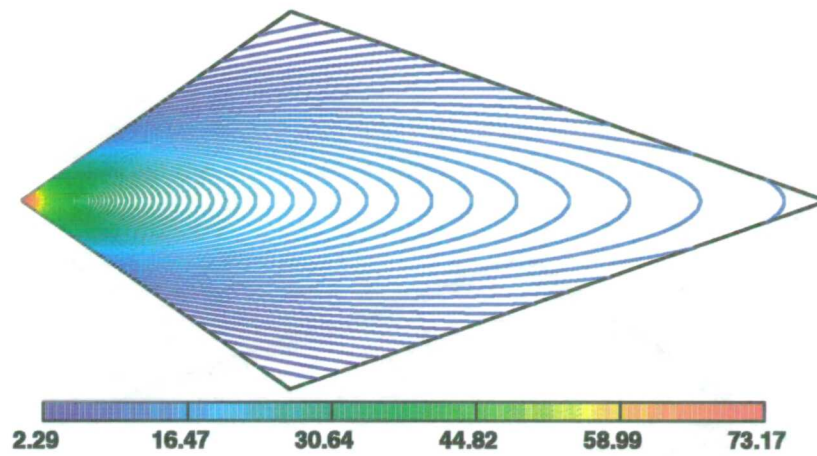


Figure 4.14: Non-aligned mesh: exact solution.

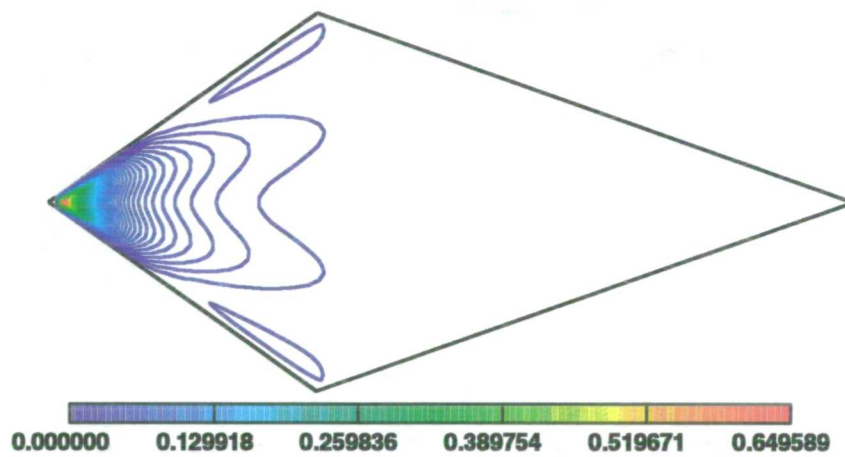


Figure 4.15: Non-aligned mesh: Exact error magnitude.

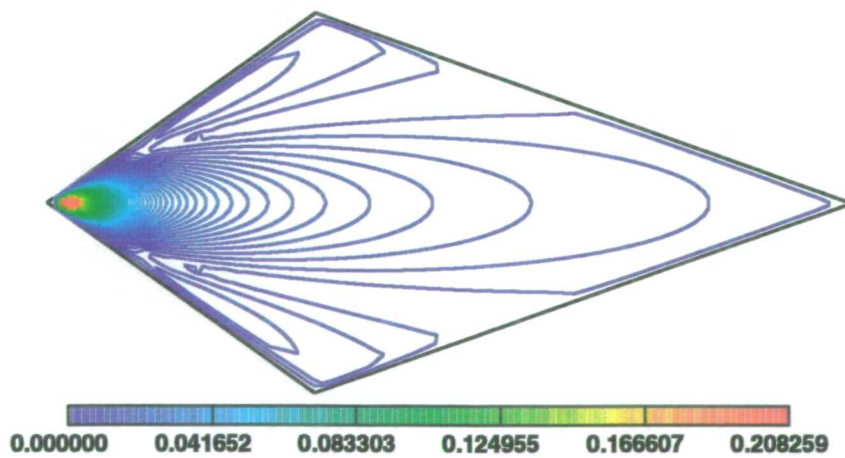


Figure 4.16: Non-aligned mesh: Direct Taylor Series Error estimate.

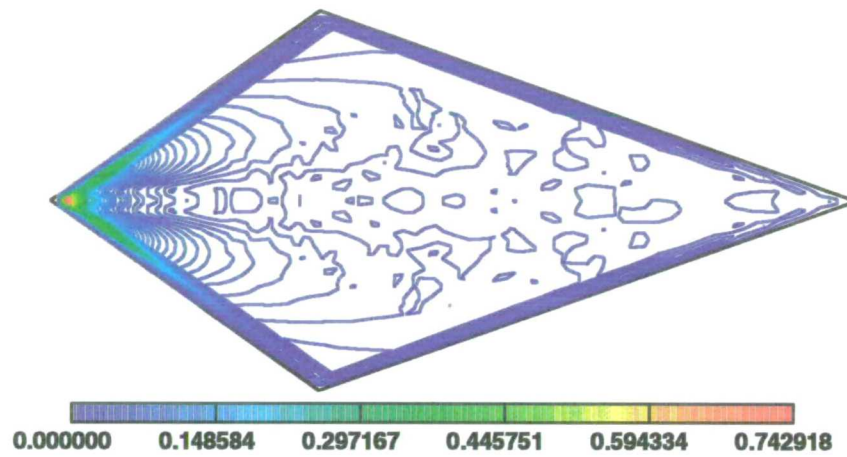


Figure 4.17: Non-aligned mesh: Moment Error estimate.

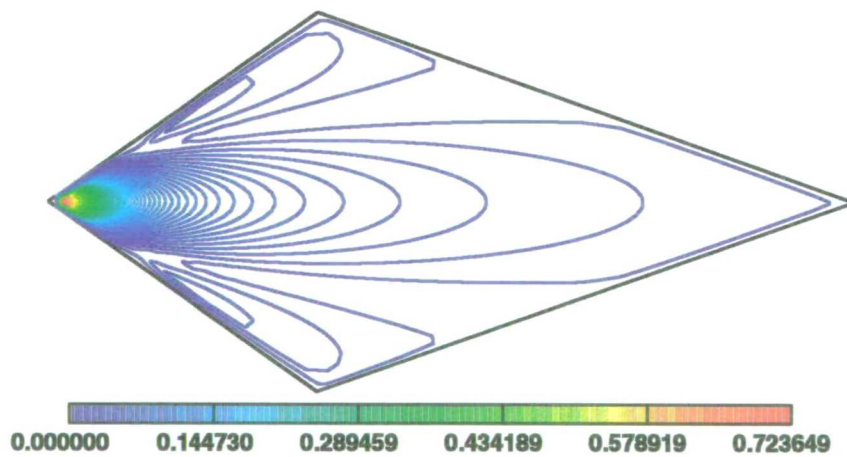


Figure 4.18: Non-aligned mesh: Residual Error estimate.

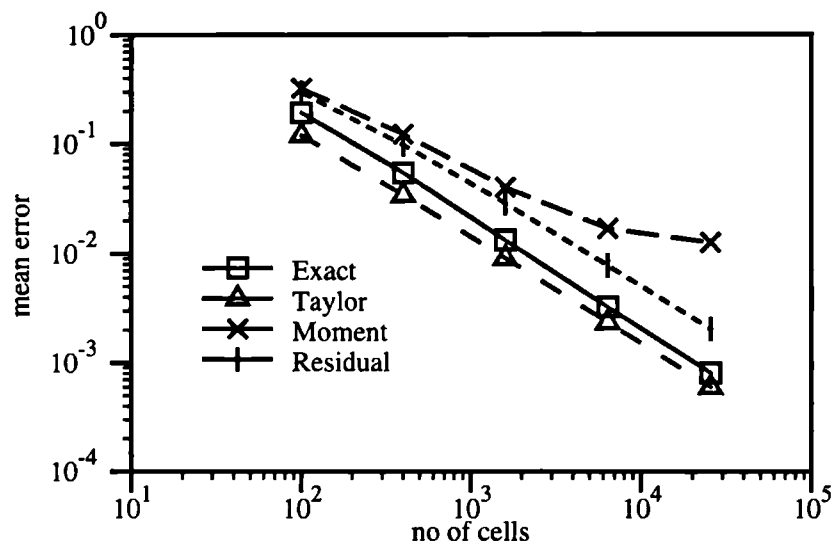


Figure 4.19: Non-aligned mesh: scaling of the mean error.

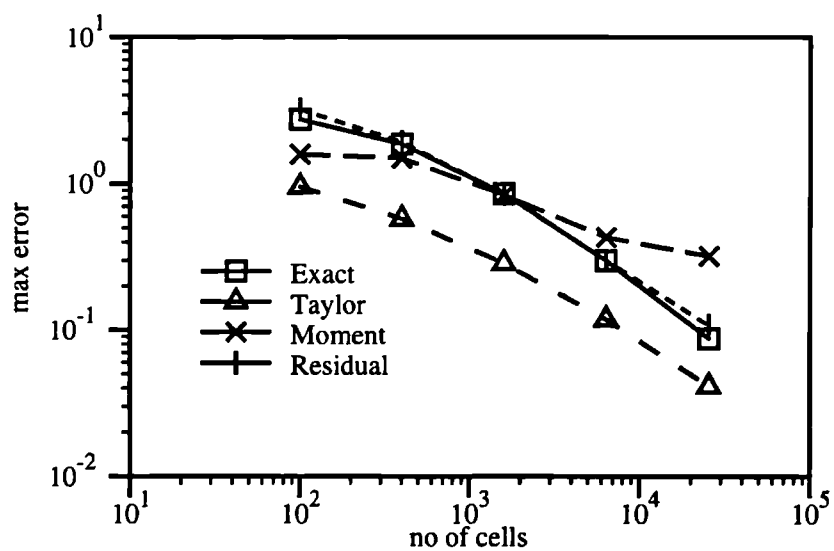


Figure 4.20: Non-aligned mesh: scaling of the maximum error.

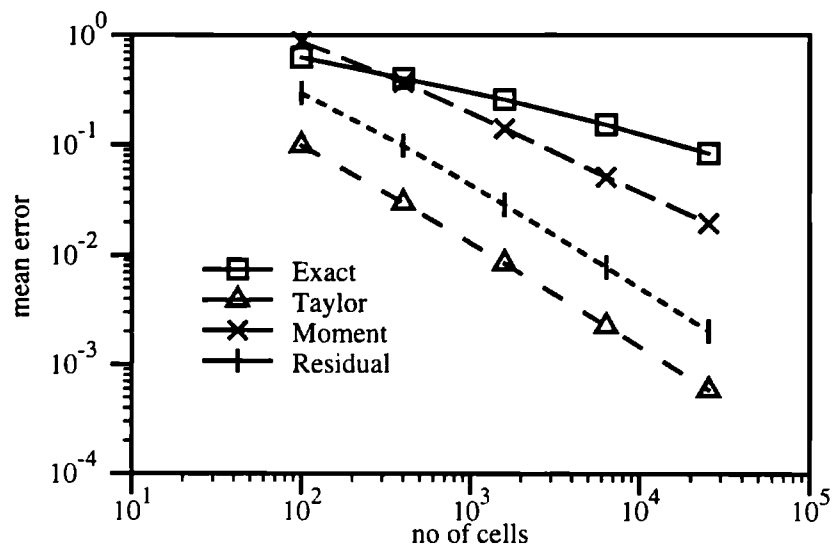


Figure 4.21: Non-aligned mesh: scaling of the mean error with UD.

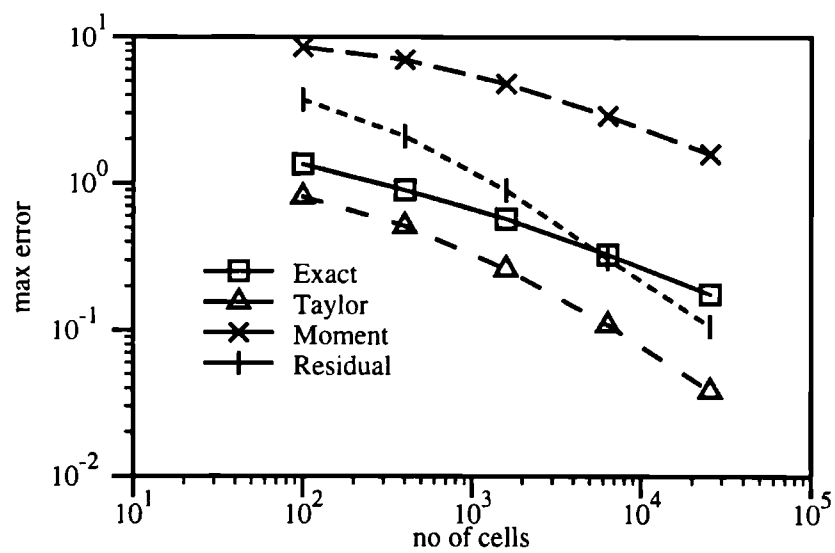


Figure 4.22: Non-aligned mesh: scaling of the maximum error with UD.

Error estimates, however, do not follow the scaling behaviour of the error. Numerical diffusion smears the gradients, producing a picture of a smoothly varying field, which complicates the estimation of the error. The quality of the solution is so bad that accurate error estimation is not possible. It should be noted that only the Moment Error estimate shows slower error reduction with refinement. This is, however, a very severe test for error estimates, with a combination of poor mesh quality, high discretisation errors and steep gradients in the solution.

4.8.2 Line Jet

In order to test the behaviour of error estimates on a fluid flow situation, a line jet test case will be considered. The test setup consists of an infinitely fast jet from an infinitely thin line orifice entering a large domain. The amount of momentum introduced by the jet is finite and equal to M . The analytical solution for this problem can be found in Panton [104]. For the coordinate system located at the mouth of the orifice with $\mathbf{i}$ pointing in the direction of the jet, the exact velocity vector in the point (x, y) is:

$$\mathbf{U} = u\mathbf{i} + v\mathbf{j}, \quad (4.126)$$

where

$$u = \frac{A}{B} x^{-\frac{1}{3}} \operatorname{sech}^2 \left(x^{-\frac{2}{3}} \frac{y}{B} \right), \quad (4.127)$$

$$v = -\frac{1}{3} A x^{-\frac{2}{3}} \tanh^2 \left(x^{-\frac{2}{3}} \frac{y}{B} \right) + \frac{2}{3} x^{-\frac{4}{3}} y \operatorname{sech}^2 \left(x^{-\frac{2}{3}} \frac{y}{B} \right), \quad (4.128)$$

$$A = \left(\frac{9}{2} \nu M_j \right)^{\frac{1}{3}}, \quad (4.129)$$

$$B = \left(\frac{48 \nu^2}{M_j} \right)^{\frac{1}{3}} \quad (4.130)$$

and $M_j = U_0^2 h$ is the momentum carried by the jet.

In order to avoid the problems with accuracy of the discretised form of the exact boundary condition, the computational domain will be located downstream from the jet. The mesh is aligned with the main direction of the flow, Fig. 4.23. The

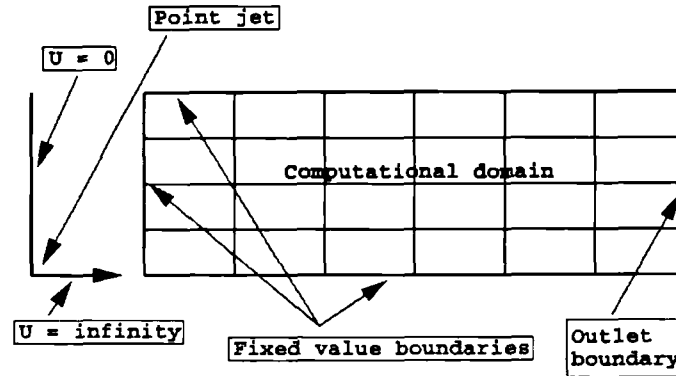


Figure 4.23: Line jet: test setup.

domain is $4 \times 1 \text{ m}$ in size, 0.5 m downstream of the orifice. Fig. 4.24 shows the exact solution on the 80×41 CV-s.

In order to simplify the discussion, only the magnitudes of the estimated and exact errors will be presented. Fig. 4.25 shows the magnitude of the latter – the region of highest error is associated with the high gradients in the vicinity of the source. In the rest of the domain, the errors are relatively low.

The proposed error estimates, Figs. 4.26, 4.27 and 4.28 pinpoint the regions of high error well. A slight anomaly in the error distribution for the Moment Error estimate can be seen in Fig. 4.27. This is a consequence of the lower order of accuracy of the Finite Volume method associated with fixed value boundaries. The face compensation of fourth-order terms of the Moment Error estimate is lost for the cells next to the boundary. These terms are large on the inlet side of the domain because of high gradients in U .

Let us now consider the scaling of the mean and maximum error for this case. Several details should be noted: on very fine meshes, the exact solution does not show expected error scaling. The error is extremely low in large regions of the mesh. Further refinement in such regions will not influence the average error. At the same time, the error is still large in the vicinity of the jet – the bulk of the solution error comes from this region. The scaling of the mean error is influenced by this feature.

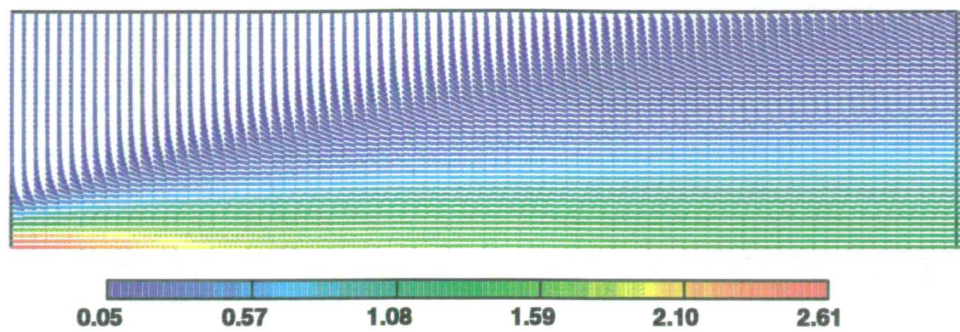


Figure 4.24: Line jet: exact solution.

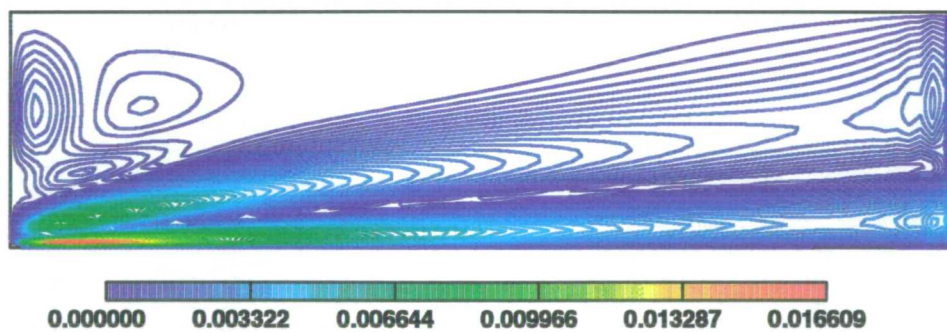


Figure 4.25: Line jet: exact error magnitude.

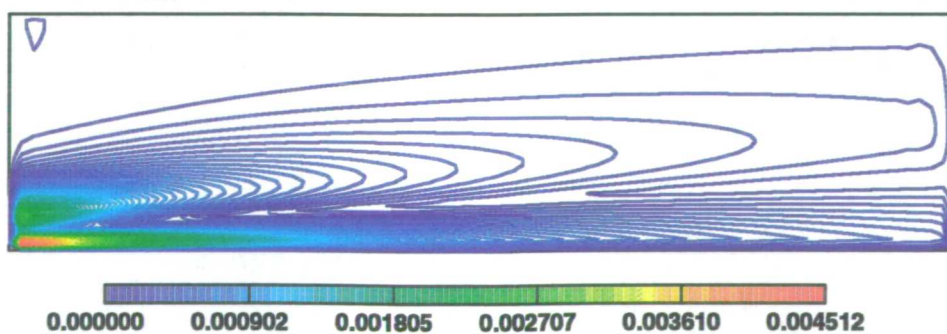


Figure 4.26: Line jet: Direct Taylor Series Error estimate.

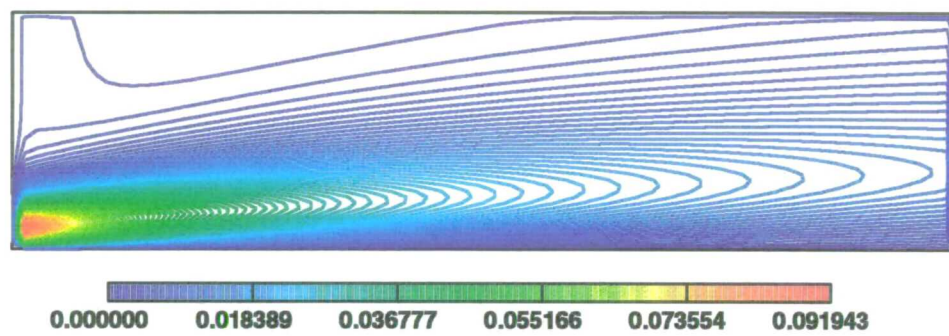


Figure 4.27: Line jet: Moment Error estimate.

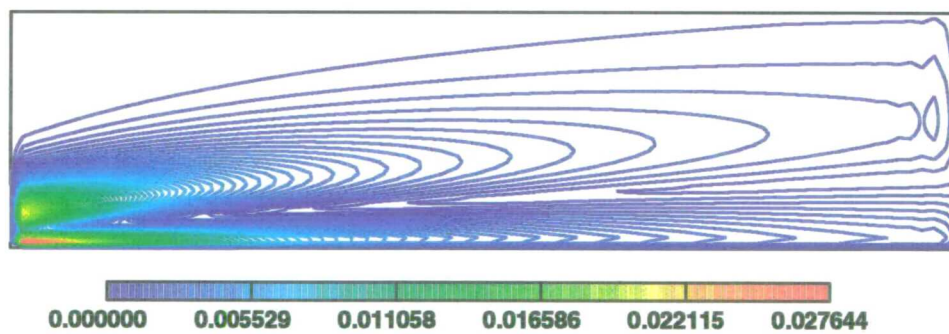


Figure 4.28: Line jet: Residual Error estimate.

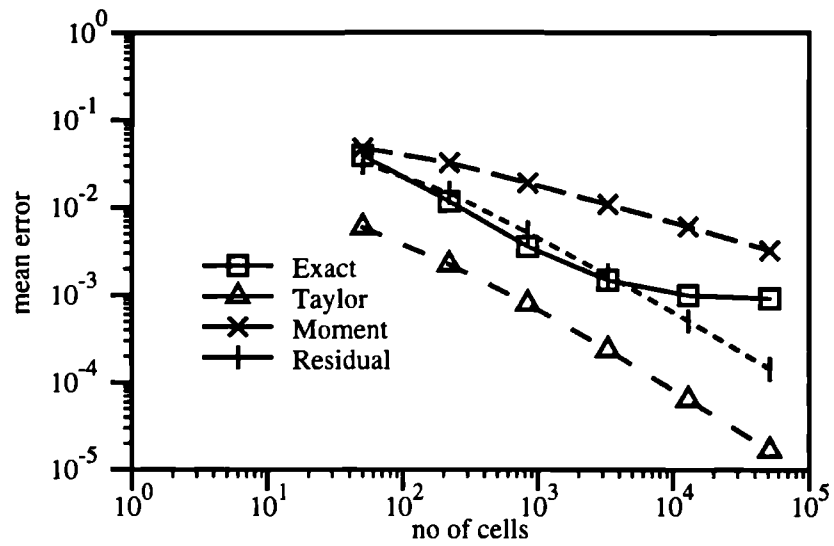


Figure 4.29: Line jet: scaling of the mean error.

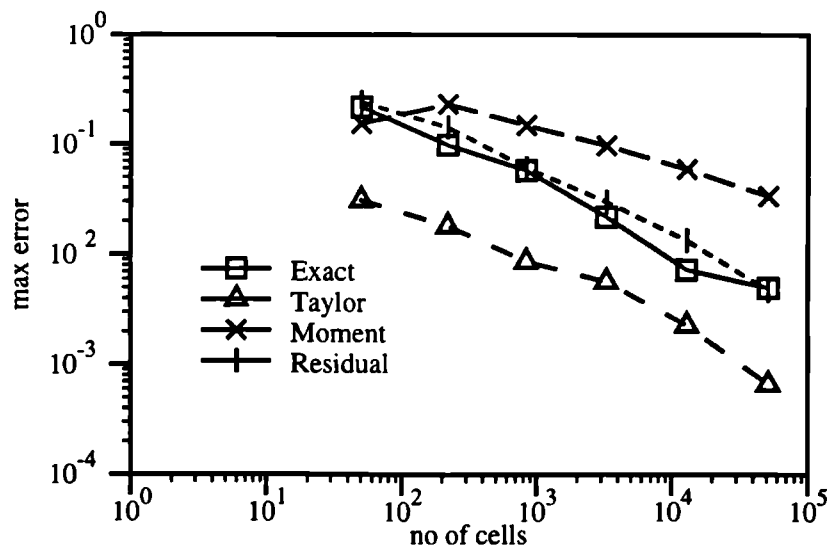


Figure 4.30: Line jet: scaling of the maximum error.

Although it shows remarkable accuracy on coarse meshes, the Moment Error estimate over-estimates both the mean and maximum error. Its error reduction rate is lower than for other methods because of the accuracy problems at the inlet boundary. The Residual Error estimate gives consistently good results. The accuracy of both methods is better than for the Direct Taylor Series Error estimate.

4.8.3 Transient One-Dimensional Convective Transport

The issue of error estimation for transient calculations will be considered in this Section. An example solution of the transport equation for the error will be given. For that purpose, the case of one-dimensional transport of a half- $\sin^2$ profile is considered. The domain is 100 CV-s long and the Courant number is set to $Co = 0.2$.

In order to create both the spatial and temporal error sources, a combination of the bounded second-order accurate convection differencing scheme (Gamma) and the second order unbounded Backward Differencing in time is used.

Fig. 4.31 shows the approximate and the exact solution after 350 time-steps. The difference between the two represents the exact cumulative error. If an estimate of this error is needed, the evolution of the error in time should be followed.

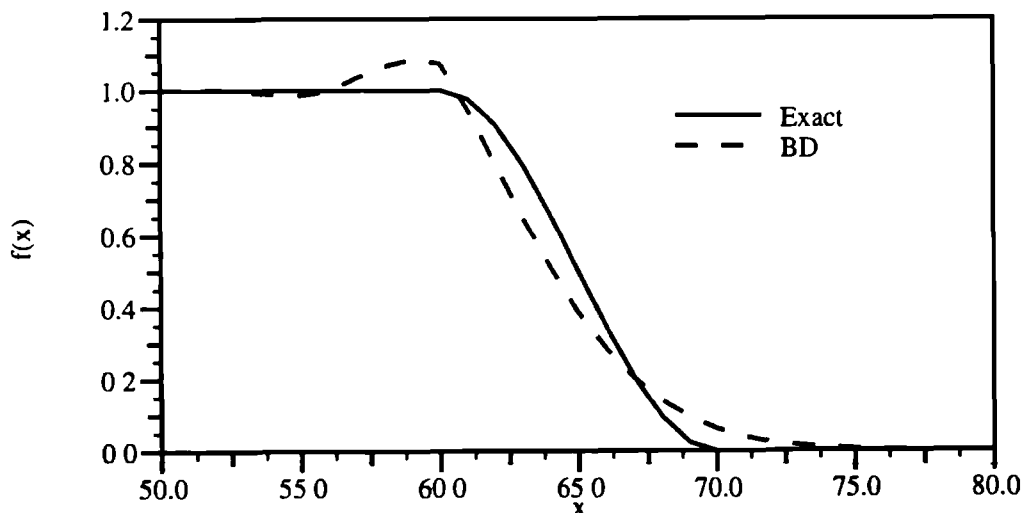


Figure 4.31: 1-D convective transport exact and analytical solution after 350 time-steps.

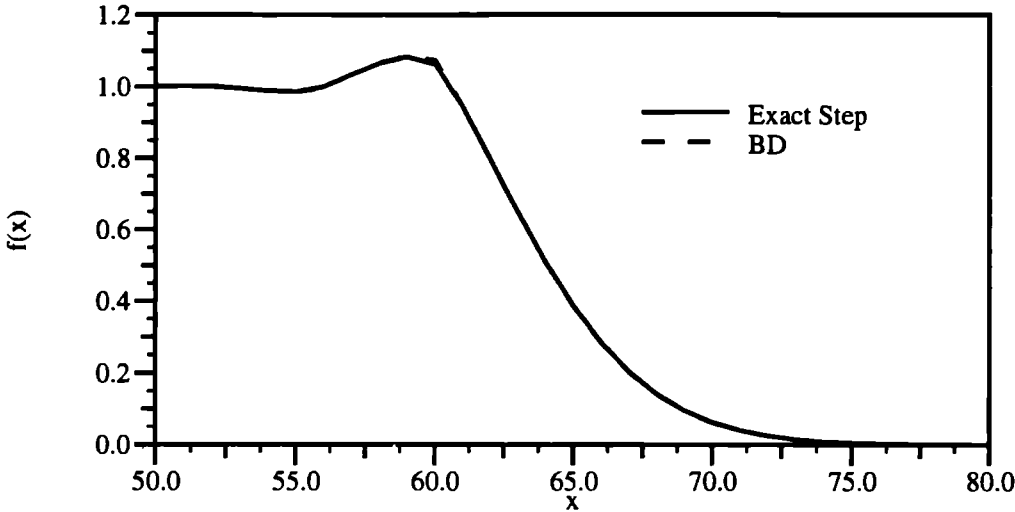


Figure 4.32: 1-D convective transport: change in the solution during a single time-step.

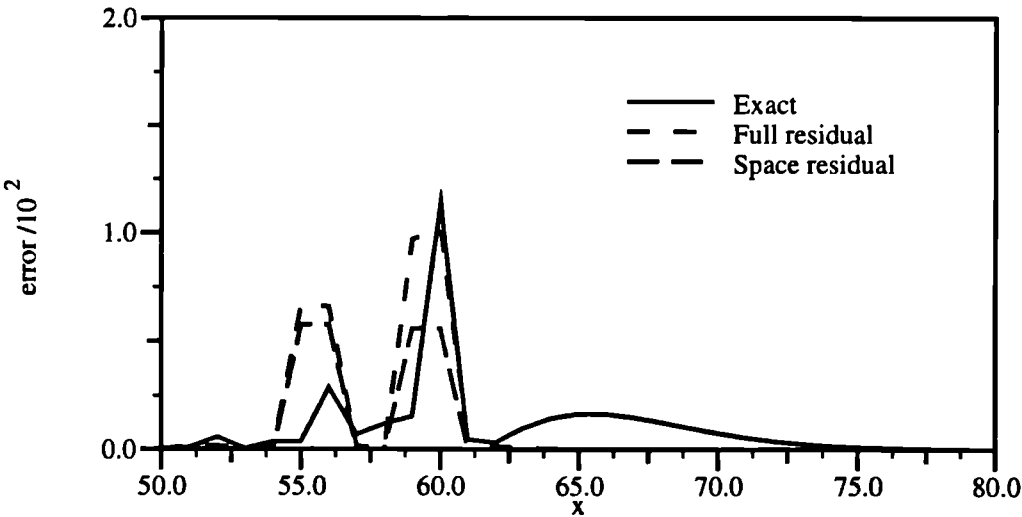


Figure 4.33: 1-D convective transport: estimated and exact single time-step error.

Let us first examine the accuracy of the error estimate for a single time-step. Comparing the shape of the solution in two consecutive steps (349th and 350th), Fig. 4.32, it is possible to determine the error introduced during this step.

Fig. 4.33 presents the exact time-step error and the full Residual Error estimate. The spatial error is included in the same graph for comparison. The full Residual Error estimate approximates the exact error well. It can also be seen that the spatial and temporal error are balanced in the region of highest error. The time-step size is therefore appropriate for the given spatial resolution.

Using the full time-step residual as the error source, it is possible to follow the evolution of the error in time from the onset of the calculation. The exact error after 350 time-steps is calculated as the difference between the two curves in Fig. 4.31. It is essential to solve the transport equation for the error as accurately as possible – in the first instance, Gamma differencing in space and Crank-Nicholson in time are used. In Fig. 4.34, the estimated and exact errors are shown. The

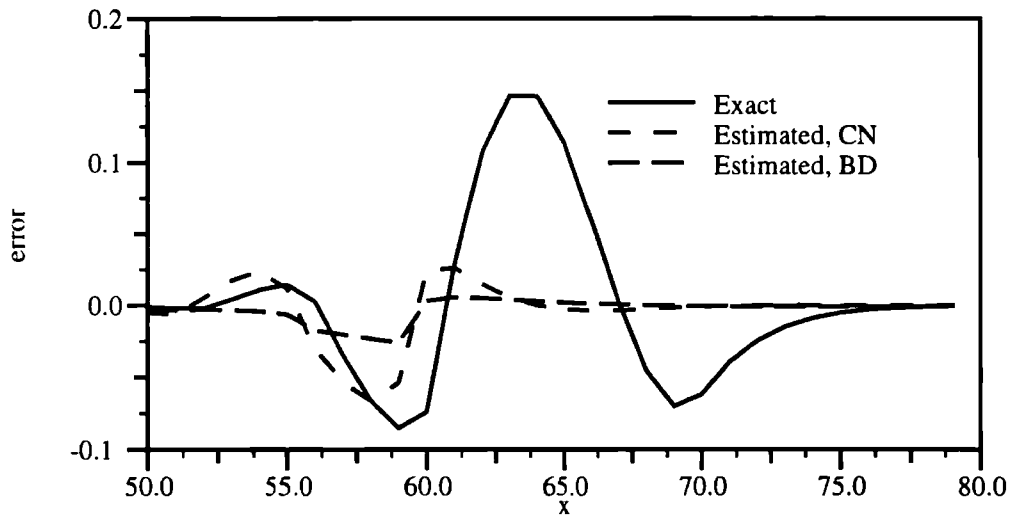


Figure 4.34: 1-D convective transport: estimated and exact error after 350 time-steps.

accuracy of this estimate is poor, as a consequence of the combined inaccuracies in the estimation of the error source terms and the actual discretisation error in the transport equation for the error. In order to determine the dependence of this inaccuracy on the discretisation of the error transport equation, another solution with Backward Differencing in time is given.

Although the estimation result for the evolution of the error cannot be considered good, it confirms the assumption about the transportive properties of the error and the residual as its source.

4.8.4 Local Problem Error Estimation

In this Section, three numerical examples for the accuracy of the Local Problem Error estimate will be presented. The accuracy of the proposed solution method for the local problem will also be examined.

In order to compare the estimated and exact error, it is necessary to evaluate the error norm from the available exact error. For each calculation, the global effectivity index will be presented.

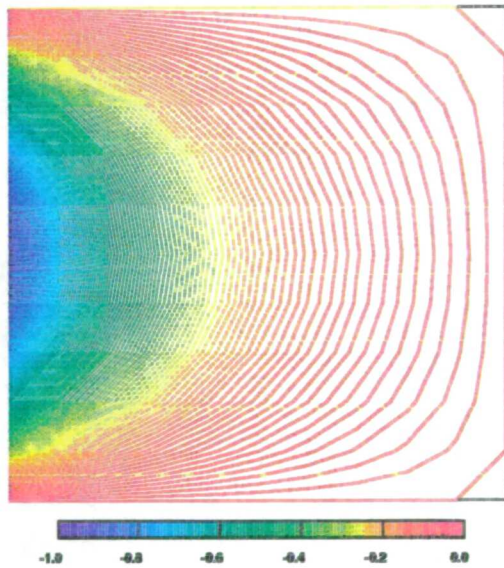


Figure 4.35: Elliptic test case: Exact solution.

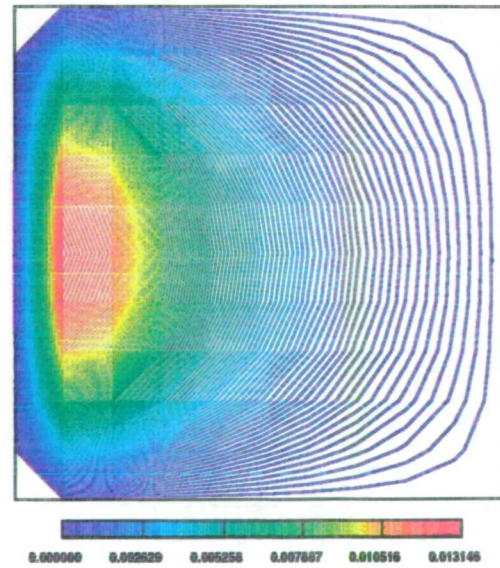


Figure 4.36: Elliptic test case: Estimated error norm distribution.

Let us start with the following elliptic test case (Ainsworth and Oden [2]):

$$-\nabla \cdot \nabla \phi = 0 \text{ in } \Omega: \left[0, \frac{1}{2}\right] \times \left[0, \frac{1}{2}\right], \quad (4.131)$$

with boundary conditions:

$$\begin{cases} \phi(0, y) = (e^{\sqrt{1+4\pi^2}} - 1) \sin(2\pi y), \\ \phi(\frac{1}{2}, y) = \phi(x, 0) = \phi(x, \frac{1}{2}) = 0. \end{cases} \quad (4.132)$$

The exact solution of the problem is:

$$\phi(x, y) = e^{(x-1)\sqrt{1+2\pi^2}} - e^{-x\sqrt{1+4\pi^2}} \sin(2\pi y). \quad (4.133)$$

The exact solution on a 10×10 mesh and the estimated error norm are shown in Figs. 4.35 and 4.36, respectively.

Exact and estimated global error norms are used to calculate the global effectivity index:

$$\zeta = \frac{e_{\|est}}{|e|_{exact}} = \frac{0.052461605}{0.0513568} = 1.021. \quad (4.134)$$

Let us now consider the accuracy of the solution procedure for the local problem. For that purpose, two control volumes will be selected – one for the determinate form of the local problem (*i.e.* with a face on the fixed value boundary) and one for the indeterminate form. The boundary conditions are calculated from the balanced error fluxes, as explained in Section 4.6. The local problem is then solved on a series of meshes and the solution is compared with the result of the simplified calculation. The results are given in Tables 4.1 and 4.2, respectively. Having in mind a very high

Mesh size	Local error norm, 10^{-6}
5×5	1.73813
10×10	1.74598
20×20	1.74822
100×100	1.74896
Approximate solution	2.12916

Table 4.1: Determinate local problem: accuracy of the approximate solution.

cost associated with a more accurate (numerical) solution procedure for the local

Mesh size	Local error norm, 10^{-5}
5×5	4.80221
10×10	4.77243
20×20	4.81892
100×100	4.85736
200×200	4.86252
Approximate solution	4.86361

Table 4.2: Indeterminate local problem: accuracy of the approximate solution.

problem, the accuracy of the simplified method seems to be acceptable, particularly because the number of cells with fixed value boundary faces is limited. The results will slightly deteriorate for the convection-diffusion local problems, but this does not seem to be critical.

Error norms for the line source in cross flow (Section 4.8.1.1) and the line jet test cases (Section 4.8.2) are shown in Figs. 4.37 and 4.38. Effectivity indices on

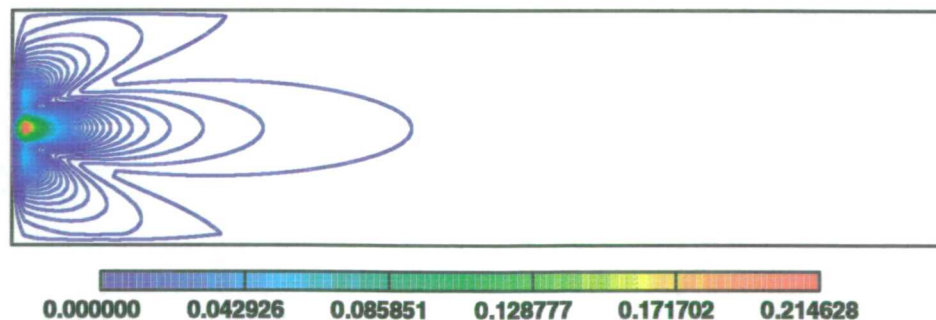


Figure 4.37: Line source in cross flow, aligned mesh: estimated error norm.

several meshes are given in Tables 4.3 and 4.4, respectively.

Both error norms pick up the region of the high exact error well. In terms of effectivity indices, the Local Problem Error estimate is superior to all other methods of error estimation. In some cases, the exact error norm has been slightly underestimated, thus violating the derived upper bound condition for the estimate. This

Mesh size	Global error norm	Global effectivity index
10×5	4.56203	1.32034
20×11	2.32105	1.10746
40×21	1.219	0.95054

Table 4.3: Line source in cross flow, aligned mesh: global error norm and effectivity index.

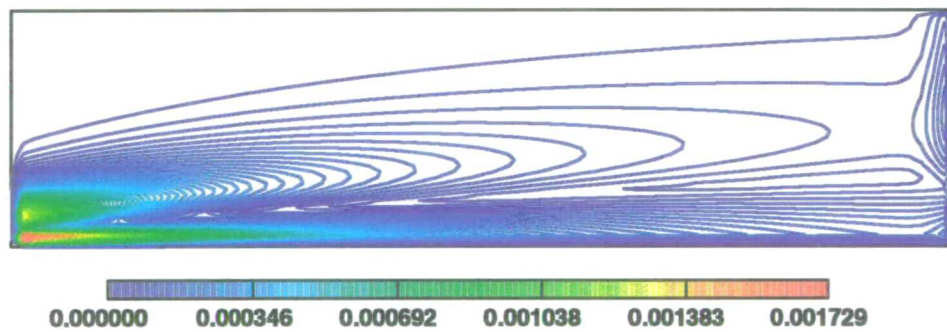


Figure 4.38: Line jet: estimated error norm.

Mesh size	Global error norm	Global effectivity index
10×5	0.393599	0.78347
20×11	0.319659	0.98313
40×21	0.017713	0.96943

Table 4.4: Line jet: global error norm and effectivity index.

is attributed to the inaccuracies in the approximate solution of the local problem.

4.9 Closure

This Chapter presented an overview of error estimation methods for the Finite Volume discretisation. Three new error estimates have been proposed. The Direct Taylor Series Error estimate is based on the analysis of the Taylor series truncation error. It is set up in such a way to enable single-mesh single-run error estimation. Its accuracy is comparable with Richardson extrapolation. The Moment Error estimate uses the imbalance in higher moments of the variable to provide an estimate of the absolute error level. The Residual Error estimate measures the error using the cell residual, caused by the inconsistency between the calculation of cell integrals and face interpolation. Appropriate normalisation procedures are used to estimate the error magnitude.

The definition of the residual for Finite Volume discretisation enables us to generalise some results from the Finite Element field. The Local Problem Error estimate has been extended to the Finite Volume method, together with the necessary error flux balancing procedure, described in Section 4.6.1.1. This method provides a highly accurate upper bound on the solution error in the energy norm. The Local Problem Error estimate, originally derived for elliptic equations has been subsequently extended to the convection-diffusion and Navier-Stokes problem.

An extension of the Residual Error estimate to transient calculations has been presented in Section 4.7. It is possible to separately estimate the parts of the error coming from spatial and temporal discretisation, as well as the total discretisation error per time-step. Issues of solution accuracy after a given number of time steps and the potential accumulation of the error have also been addressed. Finally, in Section 4.8, numerical examples illustrating the accuracy of proposed error estimates are given.

Chapter 5

Adaptive Local Mesh Refinement and Unrefinement

5.1 Introduction

Chapter 4 presented the tools that enable us to estimate the error in the numerical solution. This information can now be used to change the distribution of computational points in order to improve the accuracy of subsequent solutions.

The mesh adaptation decreases the mesh-induced discretisation errors by improving the distribution of computational points. An **automatic error-based adaptive mesh generation** algorithm tries to create a computational mesh in which the numerical error is uniformly distributed through the domain and corresponds to the desired solution accuracy. This strategy produces a solution of desired accuracy with the minimum number of computational points. It consists of several steps:

1. The geometry of the domain and the boundary conditions are provided as a set of surface descriptions and the appropriate specification of boundary conditions along these surfaces. The required level of accuracy is defined as the acceptable level of mean and maximum numerical error.
2. An initial computational mesh is created. The quality of this mesh is not

critical. It should, however, be fine enough to provide a good basis for *a-posteriori* error estimation. A poor initial mesh will increase the number of mesh adaptation cycles.

3. The discretised set of governing equations is solved on the available mesh.
4. Using the tools described in Chapter 4, the numerical errors are estimated from the information about the mesh and the available solution.
5. If the desired level of accuracy is reached, the mesh adaptation procedure is stopped.
6. It is now possible to study the interaction between the numerical error and the distribution of computational points. If necessary, additional computational points can be introduced into the mesh in order to reduce the numerical error to the desired level. As proposed by Zienkiewicz [156], the known order of accuracy of the method can be used to estimate the desired local mesh size. The new mesh is expected to have uniform error distribution.
7. A new computational mesh is created according to the criteria defined in the step 6 and the original boundary description. In order to increase the efficiency of the method, the existing numerical solution is mapped onto the new mesh and used as an initial guess.

Steps 3 to 7 are repeated until the desired level of accuracy is reached or available computer resources are exhausted.

The proposed adaptive mesh generation procedure tends towards the optimal distribution of computational points – the requirement for uniform distribution of the error is built into step 6. The most difficult part of the procedure is the automatic creation of the new mesh from the known boundary description and desired mesh size distribution. Although this seems to be a reasonable request, presently available automatic mesh generators are not able to produce meshes of reasonable quality.

An alternative approach to the subject of mesh adaptation comes from the requirement to make the procedure automatic. It is restricted by the issue of mesh

generation: instead of creating a completely new mesh, the existing mesh will be modified. Although this approach does not in general produce optimal meshes, it attempts to maximise the increase of numerical accuracy per computational cost between two mesh changes. The mesh changes are done locally, in order to preserve the “good” parts of the original mesh. If it can be assumed that the original mesh gives a good representation of the geometry of the domain and boundary conditions, the boundary description is not needed. This leads us towards an **adaptive local mesh refinement and unrefinement procedure**, consisting of the following steps:

1. An initial computational mesh is created. It is expected that this mesh is fine enough to describe the geometry and boundary conditions well¹.
2. The discretised set of equations is solved on the available mesh.
3. The numerical error is estimated. If the desired level of accuracy is reached, the calculation is stopped.
4. The estimated errors are used in order to decide what changes should be made in the mesh in order to remove the regions of high error. At the same time, it is possible that in some regions of the mesh, errors are much lower than average. A required change of the mesh therefore consists of two parts: regions of the mesh that need more computational points and regions in which the number of computational points is too large according to the ratio of the local and average error.
5. The mesh is then modified according to the requirements from the step 4. Some additional changes in the mesh may be introduced in order to preserve

¹In engineering calculations, this is rarely the case, as the boundary description consists of a set of flat faces. It is therefore necessary to update the position of the points added on the boundary in order to preserve the smooth description of the surface at the discrete level. This, on the other hand, implies the availability of an alternative surface description, either in the analytical form, or as a set on non-uniform rational b-splines NURBS, which brings us back to the interaction of the mesh refinement algorithm and the CAD package. This is not the subject of the present study and is therefore avoided.

the mesh quality.

6. The existing numerical solution is mapped on the modified mesh and used as an initial guess.

Steps 2 to 6 are repeated until the desired level of accuracy is reached.

In order to complete the adaptive local mesh refinement and unrefinement procedure, three questions need to be answered:

- How to select the regions of the mesh for refinement and unrefinement if the error distribution is known?
- How to introduce the required change in the mesh and what additional changes are needed in order to preserve the mesh quality?
- How to map the numerical solution from one mesh to the other with minimum degradation of the solution accuracy?

The first question is answered in Section 5.2 refinement and unrefinement criteria are based on the analysis of the estimated error field. An answer to the second question is proposed in Section 5.3. The mesh changes are based on an embedded cell-by-cell refinement and unrefinement procedure for “1-irregular” meshes. Some details of the solution transfer between the meshes are given in Section 5.4. In order to provide an insight into the relative merit of the proposed refinement technique in comparison with uniform refinement, the error reduction on a simple test case is presented in Section 5.5. Finally, the adaptive local mesh refinement and unrefinement algorithm is summarised in Section 5.6, together with some closing remarks.

5.2 Selecting Regions of Refinement and Unrefinement

The procedure of selecting the regions of the mesh for refinement and unrefinement is influenced by two issues: the mesh adaptation procedure that will be used and

the accuracy of the error estimate. The required information can also be two-fold. Adaptive mesh generation algorithms need a distribution of the desired mesh size through the domain, whereas local mesh refinement requires the lists of cells labelled for refinement and unrefinement. Although these lists are related to the desired mesh size distribution, it is easier to adopt a selection procedure rather than try to transform one into the other.

Let us first show how to select the desired mesh size distribution. For each cell of the computational mesh, the error magnitude and the mesh size h are known. It is also known that the discretisation method is second-order accurate, *i.e.* that the error scales with the square of the mesh size. If the desired error level is E_0 , the desired local mesh size l_0 is calculated from the following scaling relations for the error:

$$|e| \sim A h^2, \quad (5.1)$$

$$E_0 \sim A l_0^2, \quad (5.2)$$

where A is the unknown scaling factor. It follows that:

$$l_0 = h \sqrt{\frac{E_0}{|e|}}. \quad (5.3)$$

The quantity l_0 is calculated for each computational point. The field of l_0 can subsequently be smoothed out in order to preserve reasonable mesh grading.

The local mesh refinement procedure can in theory use the same result. The local value of l_0 is compared with the existing mesh spacing in order to decide whether the cell needs to be refined or unrefined. Unfortunately, this is not the best approach in some cells, the desired mesh size can be considerably smaller than their current size. If the mesh quality is to be preserved, grading between the fine and coarse regions needs to be smooth.

As an alternative approach, the error in a cell can be compared with the average error $\bar{e}$ in the domain. Refinement and unrefinement criteria are then:

- If the error is larger than $\lambda_{ref} \bar{e}$, the cell is marked for refinement.
- If the error is smaller than $\lambda_{unref} \bar{e}$, the cell is marked for unrefinement.

Parameters λ_{ref} and λ_{unref} are the “safety” coefficients controlling the refinement procedure, with the following properties:

$$\lambda_{ref} > 1, \quad (5.4)$$

$$\lambda_{unref} < 1 \quad (5.5)$$

Although this system produces the refinement information efficiently, it is by no means as accurate as the first approach. For example, in early stages of refinement starting from a coarse mesh, the error level may be high everywhere and most of the cells need to be refined. If, on the other hand, the solution is known to have some localised features (shocks, propagating discontinuities *etc.*), a high level of localised refinement is needed. Only a very small number of cells needs to be changed.

Parameters λ_{ref} and λ_{unref} effectively control the shape of the final mesh. Higher λ_{ref} and lower λ_{unref} result in a larger number embedded levels of refinement. Ideally, λ_{ref} and λ_{unref} should be based on the statistics of the error distribution, average error level and the required accuracy. Unfortunately, no simple relation between these parameters can be found. The recommended values of refinement and unrefinement control parameters based on the experience in this study are:

$$\lambda_{ref} = 1.5, \quad (5.6)$$

$$\lambda_{unref} = 0.5. \quad (5.7)$$

Lists of cells for refinement and unrefinement do not contain all the information needed to construct a good locally refined mesh. It is still desirable to create a mesh with a smooth transition between the coarse and fine regions. The grading information is not included – smoothness of the mesh needs to be guaranteed in some other way. One of the possibilities is to create a set of rules describing the interaction between the coarse and fine regions. Cells for refinement (unrefinement) are marked and the refinement information is propagated through the mesh. A more detailed discussion of this issue will be given in Section 5.3. Another possibility is to group the cells that require refinement into blocks and then add a “safety margin” around each block (Brandt [22], Caruso [24, 25]). This moves the coarse-fine mesh interface away from the region of high error. Safety regions should in

some way take into account the distribution of desired mesh spacing around the regions of high error. This brings us to the issue of accuracy of the error estimate. Considering the potentially high grading of locally refined meshes, it is important that the error estimate represents the error accurately. Safety regions are introduced to compensate for the potential inaccuracy of the estimate. Generally, an accurate error estimate allows a precise (and therefore efficient) mesh refinement procedure. The accuracy of the estimate is more important in the local refinement procedure than in adaptive mesh generation, where mesh grading is naturally smoother.

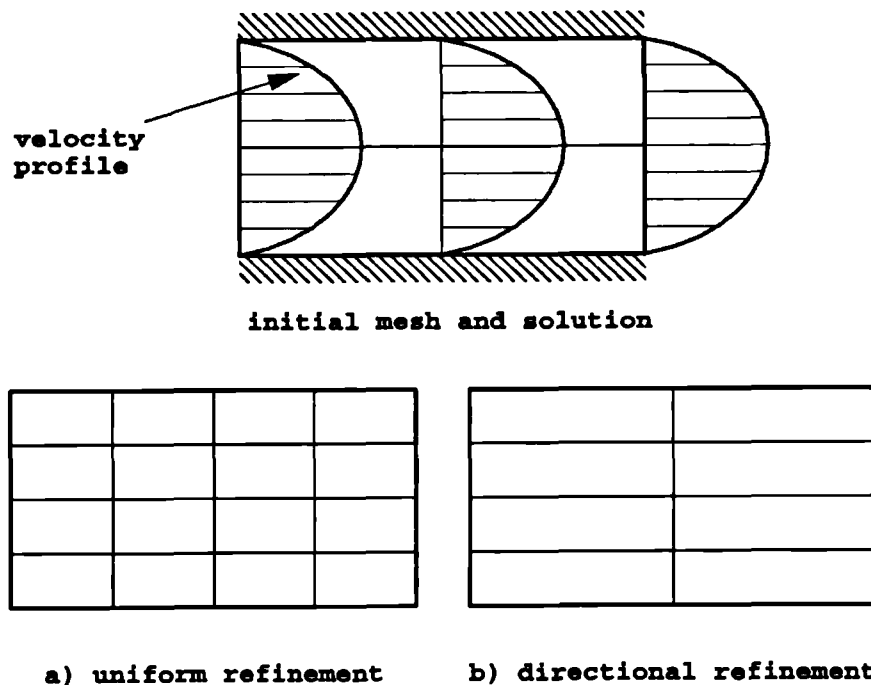


Figure 5.1: Directionality of mesh refinement.

The last issue that needs to be addressed is the local orientation of the mesh. Let us consider a fully developed laminar duct flow, Fig. 5.1. The velocity profile changes rapidly across the duct, but remains uniform along the duct. It would therefore be reasonable to refine the mesh only in the direction normal to the gradient (Fig. 5.1 b) and not equally in all directions (Fig. 5.1 a).

The directionality information can be deduced from the gradient of the solution, which points in the direction of the steepest slope. It is therefore desirable to refine the mesh preferentially normal to the gradient vector.

If the computational procedure includes the solution of more than one equation, cells for refinement are combined into a single list. It may happen that a single cell is marked for refinement by more than one error field. In that case, the desired refinement direction needs to take into account the gradients of all fields requiring refinement of that particular cell.

The implementation of refinement directionality is easier in the context of local mesh refinement than in adaptive mesh generation and will be further discussed in the next Section.

5.3 Mesh Refinement and Unrefinement

Mesh refinement and unrefinement changes the topology of the mesh in order to include the regions of refinement and unrefinement. The available information includes:

- Current computational mesh,
- List of cells that need to be refined with the desired direction of refinement,
- List of cells for unrefinement.

Let us first establish a refinement procedure for a single cell. A decision on the type of the cell split needs to be made. In order to preserve the quality of the mesh, the available cell types are limited to hexahedra and degenerate hexahedra.

Consider a hexahedral cell and a refinement direction $\mathbf{R}$, Fig. 5.2. The shape of the cell is described by three vectors: $\mathbf{1}$, $\mathbf{2}$ and $\mathbf{3}$. They carry the information about the directional stretching of the control volume. These vectors have an average direction of the two face area vectors for the pair of opposite faces of the hexahedron and the magnitude corresponding to the ratio of the volume and the average cross-section area.

For example, vector $\mathbf{1}$ is calculated as:

$$\mathbf{1} = \left(\frac{\mathbf{S}}{S} + \frac{\mathbf{S}_1}{S} \right) \frac{V_P}{S + S}, \quad (5.8)$$

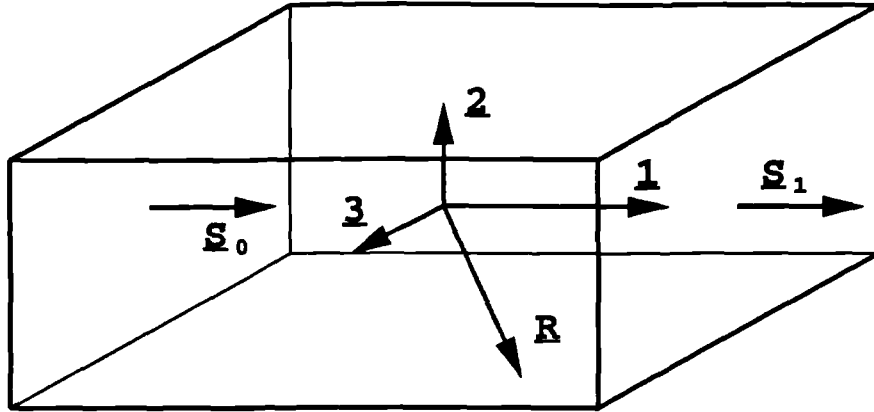


Figure 5.2: Refining a hexahedral cell.

where V_P is the cell volume and S_0 and S_1 are the face area vectors for the pair of opposite faces, Fig. 5.2. Vectors **2** and **3** are calculated equivalently.

The decision on the cell split is based on the magnitude of the dot-product of **1**, **2** and **3** with **R**. The importance of each direction is estimated as:

$$\tau_1 = |\mathbf{R} \cdot \mathbf{1}|, \quad (5.9)$$

$$\tau_2 = |\mathbf{R} \cdot \mathbf{2}|, \quad (5.10)$$

$$\tau_3 = |\mathbf{R} \cdot \mathbf{3}| \quad (5.11)$$

and the average is:

$$\bar{\tau} = \frac{1}{3}(\tau_1 + \tau_2 + \tau_3). \quad (5.12)$$

Three directions are now examined separately. If τ_i for a certain direction is larger than $\xi \bar{\tau}$, the cell is split normal to that direction. This procedure takes into account the interaction between the geometry of the cell and the direction of the gradient, with ξ as a parameter controlling the “directionality” of refinement. Higher levels of ξ imply that the solution has one strong direction of the gradient and strong mesh-to-flow alignment is desired. ξ should vary between zero (cell is always split in all three directions) and two (strong directional refinement). For a good compromise between alignment and smoothness, the currently recommended values of ξ are 0.6 for two-dimensional situations and 1.2 for three-dimensional flows.

Smooth grading in locally refined meshes is a more difficult task. The freedom in grading is very small – we can decide either to split the cell or not. Smoothly graded meshes could be obtained through point movement algorithms (see Section 1.2.3), but that approach is considered to be too expensive for the mesh quality it produces. However, it can be at least ensured that the local jump in the cell size is not larger than two. In order to do that, a concept of “1-irregular” meshes (Demkowicz *et al.* [40]) is introduced. An example of such a mesh in two dimensions is shown in Fig. 5.3. In the first level of refinement, cells in the hatched region were selected

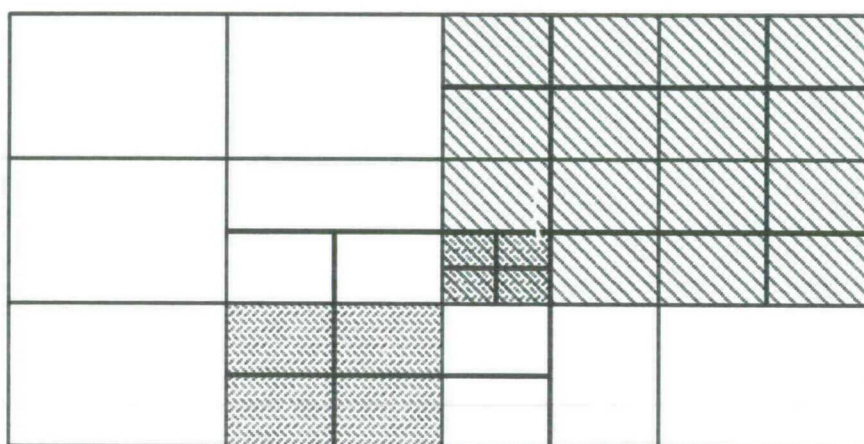


Figure 5.3: “1-irregular” mesh.

for refinement. No additional intervention was necessary. The second refinement level required the shaded cells to be split. In order to preserve smooth grading, the concept of “1-irregularity” comes into action. The new topological cell type is a “split hexahedron”. It is hexahedral in shape, but can have two “neighbours” over the one of the faces of the original hexahedron. If there is more than one such face, the cell needs to be subdivided. Additional changes in the mesh necessary to preserve “1-irregularity” were done in the two cells next to the refinement region. The resulting mesh is again “1-irregular”. As a consequence of the “1-irregularity” condition, the grading in the final mesh is smoother.

Unlike the cells marked for refinement, for which refinement is forced, cells for unrefinement carry only a possibility of being removed from the mesh. In other words, unrefinement will be performed only if there are no mesh regularity require-

ments to prevent it. A standard approach to mesh unrefinement enables the removal of only those cells that have been added during the refinement procedure. This automatically implies that the mesh refinement starts from a very coarse mesh and that the final mesh can only be locally as fine or finer than the original mesh. Cells added into the mesh are stored into a “tree” data structure and can be removed on request. An advantage of this strategy is that the original mesh can be recovered after refinement and unrefinement.

In some situations, the refinement-only approach is not appropriate. The initial mesh can be too coarse to produce a reasonable error estimate. A more general procedure, including an arbitrary initial mesh with the possibility of both refinement and unrefinement has been adopted in this study.

The unrefinement procedure is based on a “cell-pair” concept: the list of cells for unrefinement is scanned in order to create “cell pairs” according to certain criteria. The cell pair can be subsequently merged into a single cell if there are no regularity limitations. It is no longer necessary to store the “history” of the mesh refinement, as any cell is capable of unrefinement. One of the drawbacks of this approach is that the recovery of the initial mesh after refinement and unrefinement is not guaranteed. This is, however, desirable only in transient shock-tracking calculations. If mesh recovery is essential, the refinement history can be built into the original procedure.

5.4 Mapping of Solution Between Meshes

In order to speed up the solution of the problem on a refined mesh, a good initial guess is needed. An approximate solution of the same problem on a different mesh is already available, since the problem has been solved before the mesh is refined.

The mapping of the solution between two meshes is a straightforward task. The assumed variation of the function over each control volume is linear. All fields defined on cell centres can be mapped using the following procedure:

1. For each point P in the refined mesh, find the closest point T in the original

mesh.

2. Calculate the position difference:

$$\mathbf{p} = \mathbf{x}_P - \mathbf{x}_T, \quad (5.13)$$

3. Calculate ϕ_P as

$$\phi_P = \phi_T + \mathbf{p} \cdot (\nabla \phi)_T. \quad (5.14)$$

The face flux transfer requires more care. The fluxes are defined on cell faces and satisfy the continuity constraint. If the flux transfer is not accurate, continuity will be violated, possibly causing unboundedness in the first solution of the new mesh and other undesirable effects.

Since not all of the faces have their equivalents in the original mesh, a more general (and more accurate) procedure is needed. For that purpose, we need to come back to the calculation of fluxes (see Section 3.8). Fluxes are calculated using Eqn. (3.144):

$$F = \mathbf{S} \cdot \left[\left(\frac{\mathbf{H}(\mathbf{U})}{a_P} \right)_f - \left(\frac{1}{a_P} \right)_f (\nabla p)_f \right],$$

after the solution of the pressure equation, Eqn. (3.141):

$$\nabla \cdot \left(\frac{1}{a_P} \nabla p \right) = \sum_f \mathbf{S} \cdot \left(\frac{\mathbf{H}(\mathbf{U})}{a_P} \right)_f.$$

Eqn. (3.141) guarantees that the fluxes satisfy continuity. Instead of transferring the fluxes directly, parts of the pressure equation will be interpolated to the new mesh. The required fields are a_P and $\mathbf{H}(\mathbf{U})$ both defined at cell centres and easy to transfer. The pressure equation is then assembled and solved on the new mesh before the calculation is resumed. This procedure can be performed between the meshes bearing no similarity and will always produce conservative fluxes.

5.5 Numerical Example

The influence of the mesh changes introduced by the adaptive procedure to the accuracy of consequent solutions is two-fold.

- The numerical error caused by insufficient mesh resolution in the original mesh is decreased through the introduction of additional computational points in the regions of high error. If the error estimate is sensitive to the other mesh-induced errors, such as mesh skewness and excessive non-orthogonality, the cells causing the error are also subdivided, thus reducing the error magnitude².
- Local refinement by mesh embedding increases mesh non-orthogonality, which may have adverse effects on the solution accuracy. The increase in the local error is created away from the originally detected high error.

A good mesh adaptation algorithm should offer the benefit of adequate local mesh resolution without the significant deterioration in mesh quality. In order to examine the performance of the adaptive algorithm described in this Chapter, its error reduction rate will be compared with that of uniform refinement on a test case with an analytical solution. For this purpose, the line source in cross-flow test case described in Section 4.8.1.1 will be used. The influence of the error estimation method on the error reduction rate will also be examined.

In the first instance, the adaptive refinement starts from a coarse 10×6 CV mesh. The refinement parameters are set to:

$$\begin{aligned}\lambda_{ref} &= 1.5, \\ \xi &= 0.6.\end{aligned}$$

The initial mesh and first six levels of adaptive refinement based on the Residual Error estimate are shown in Figs. 5.4, 5.5, 5.6, 5.7, 5.8, 5.9 and 5.10.

The final mesh, with 10 embedded levels of refinement, is shown in Fig. 5.11. As a consequence of the symmetry of the problem, the refinement procedure creates symmetric refinement patterns. The “1-irregularity” principle successfully enforces a smooth transition between the coarse and fine regions of the mesh.

The nature of the adaptive algorithm is such that the maximum error moves to different regions of the domain, as the cells are added in the region of highest

²Only the estimates based on the Taylor series truncation error are not sensitive to mesh quality.

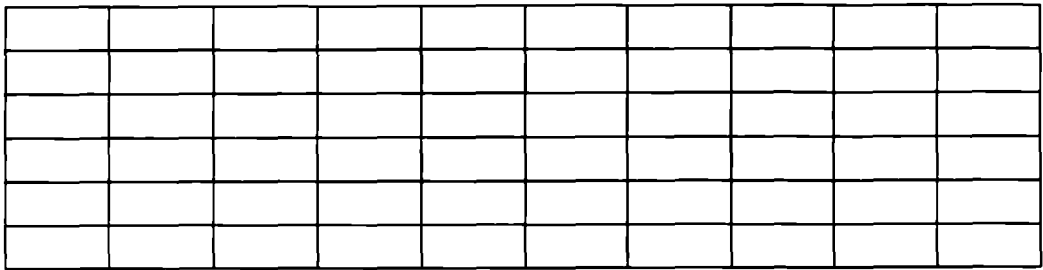


Figure 5.4: Initial mesh.

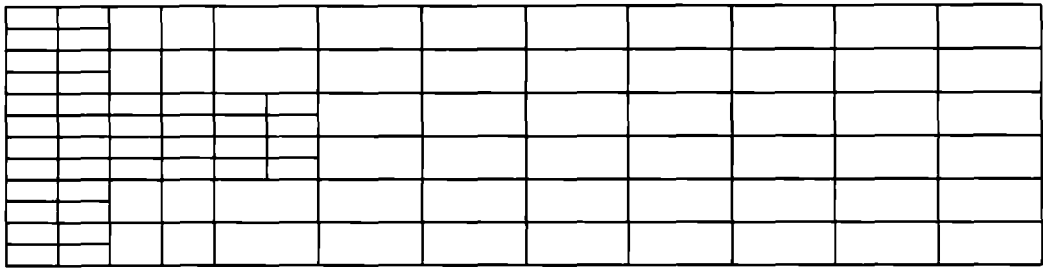


Figure 5.5: First level of adaptive refinement.

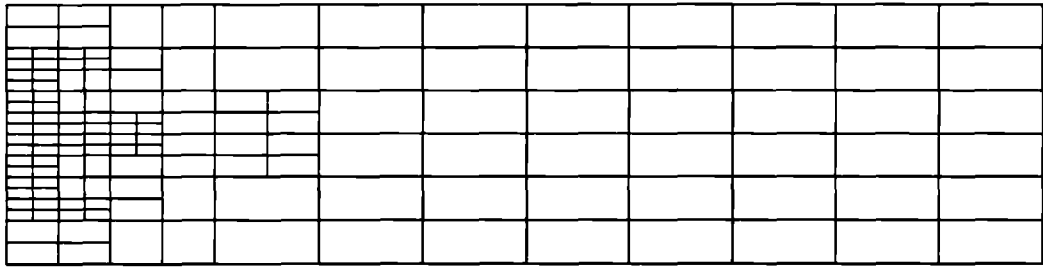


Figure 5.6: Second level of adaptive refinement.

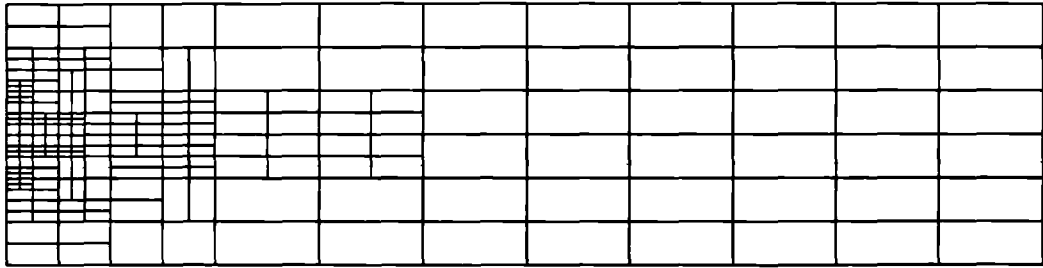


Figure 5.7: Third level of adaptive refinement.

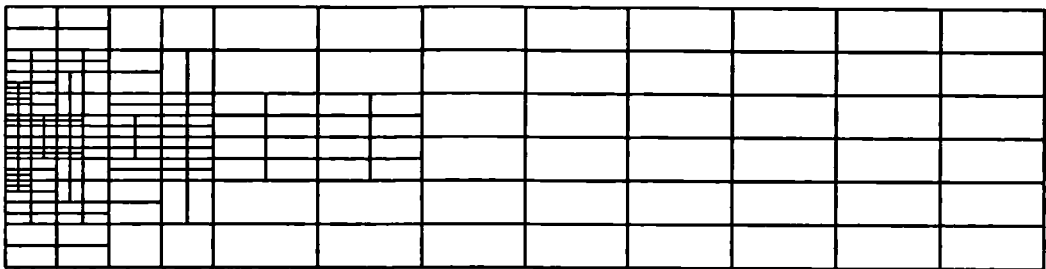


Figure 5.8: Fourth level of adaptive refinement.

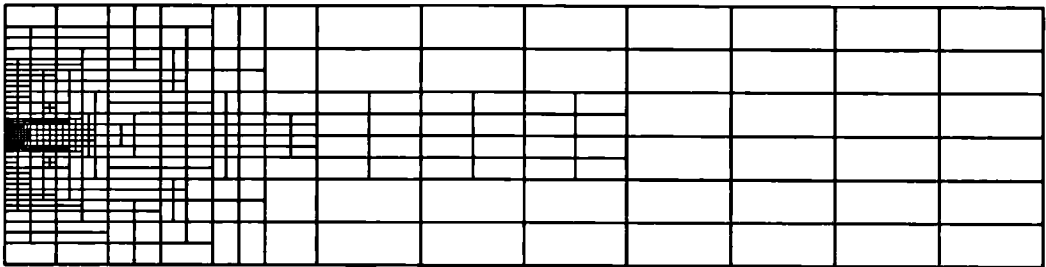


Figure 5.9: Fifth level of adaptive refinement.

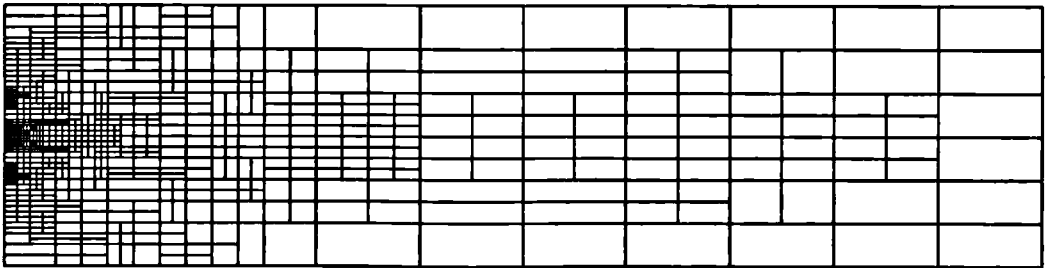


Figure 5.10: Sixth level of adaptive refinement.

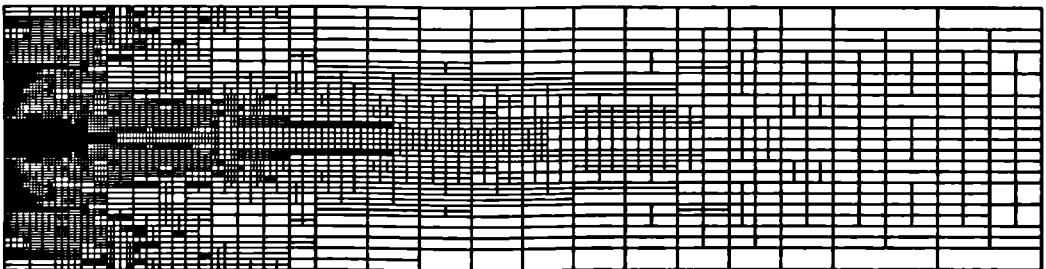


Figure 5.11: Tenth level of adaptive refinement.

error. The distribution of the exact error in the initial mesh is shown in Fig. 5.12. Figs. 5.13, 5.14 and 5.15 show the “movement” of the error after two, four and

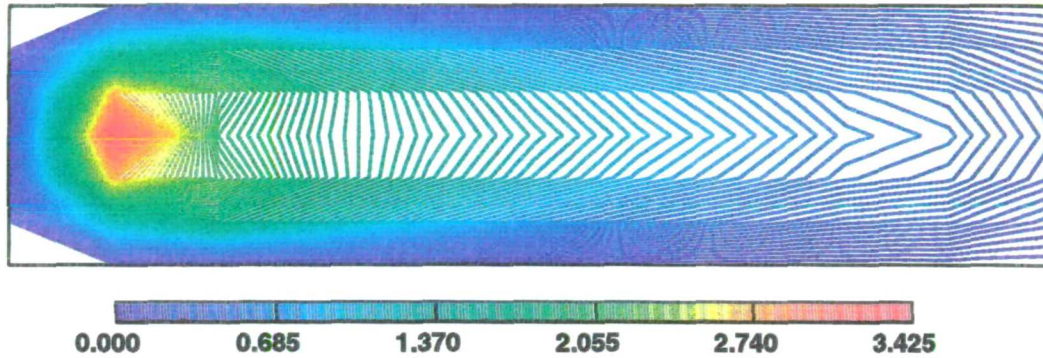


Figure 5.12: Initial distribution of the exact error.

six levels of refinement. In order to clarify the process, the scale in all figures has been changed. As the refinement progresses, the error peak is reduced, leading to a more uniform error distribution. In the case of uniform refinement, the error distribution remains similar through the whole refinement procedure (see Fig. 4.7). Following the definition of the optimal mesh from Section 4.6, the quality of the adaptively refined mesh is better than that of the uniform mesh, irrespective of the mesh-induced errors caused by mesh embedding.

The reductions of the mean and maximum error with the number of control volumes for uniform and adaptive refinement are shown in Figs. 5.16 and 5.17. In spite of the rapid reduction of the maximum error, in the case of adaptive refinement the mean error, after an initial period, decreases slower than in the case of uniform refinement. This result suggests that after an initial gain in accuracy of the adaptive procedure, the uniform refinement becomes more efficient. The mean error reduction in Fig. 5.16 should be approached with care. The mean error for both uniform and adaptive refinement is calculated as the average value for all computational points. In the case of uniform refinement, the points are uniformly distributed through the domain, covering both the regions of high and low error. Adaptive refinement, on the other hand, places the points in regions of high error, thus resolving the features that have caused the error in the first place but at the same time biasing the mean

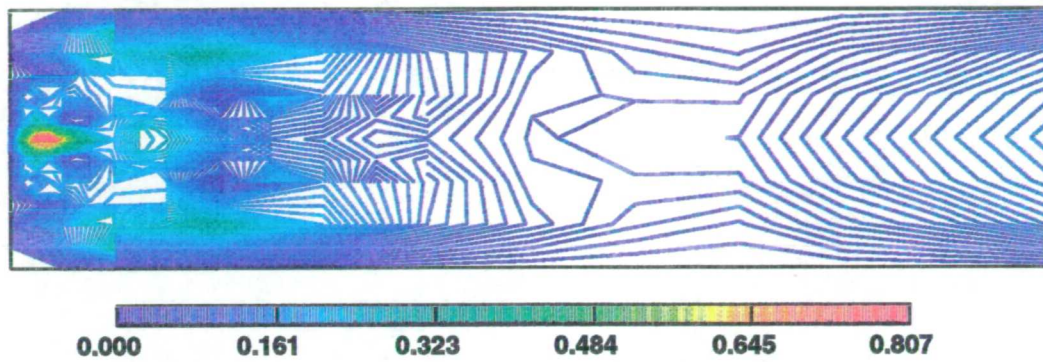


Figure 5.13: Exact error distribution after two levels of adaptive refinement.

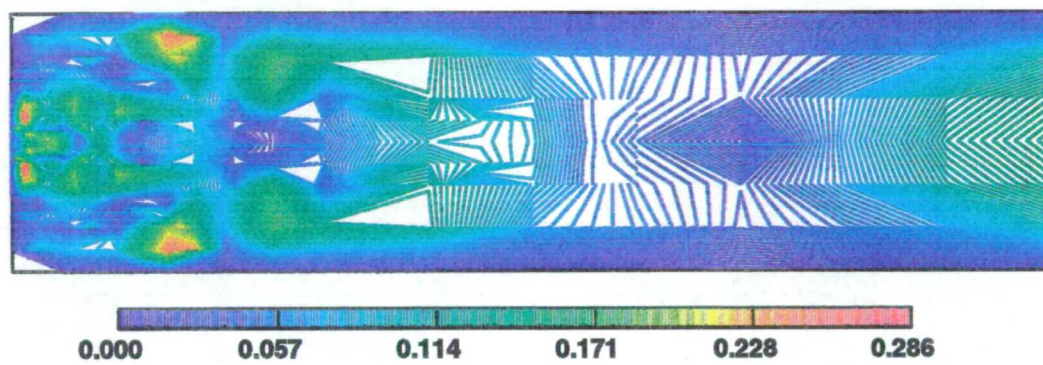


Figure 5.14: Exact error distribution after four levels of adaptive refinement.

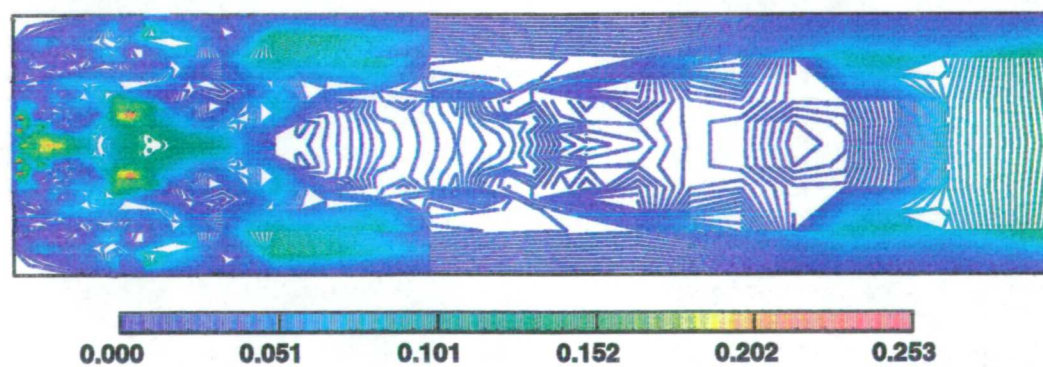


Figure 5.15: Exact error distribution after six levels of adaptive refinement.

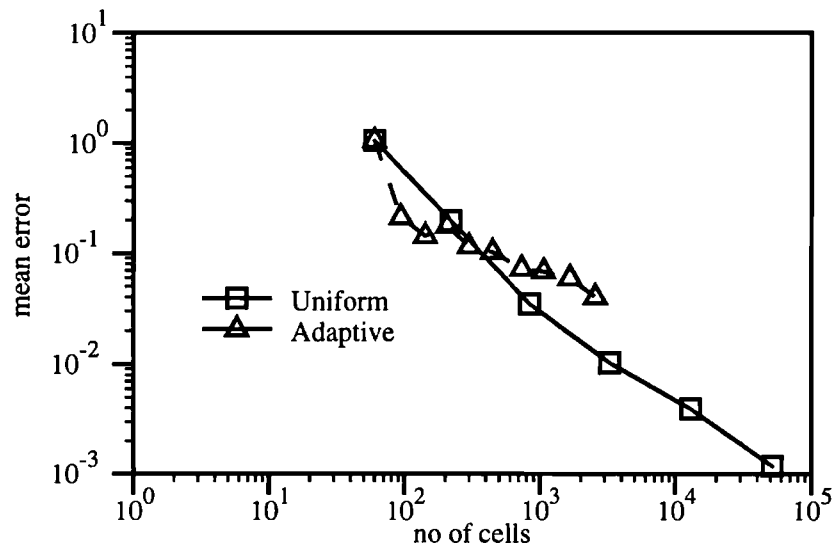


Figure 5.16: Uniform and adaptive refinement: scaling of the mean error.

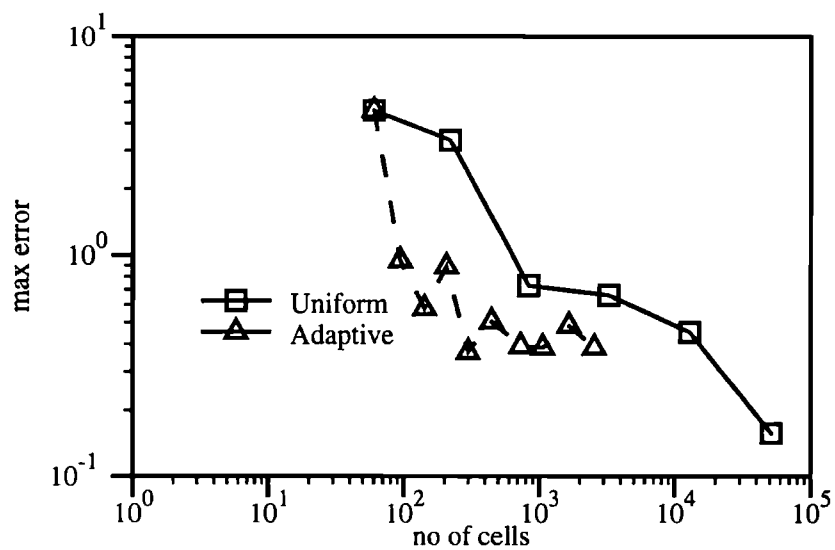


Figure 5.17: Uniform and adaptive refinement: scaling of the maximum error.

by higher sampling in those regions. This effect can be partially removed if a cell-volume-weighted average error is used as shown in Fig. 5.18, but the sampling bias still exists. Having in mind that the sampling problem cannot be resolved in an

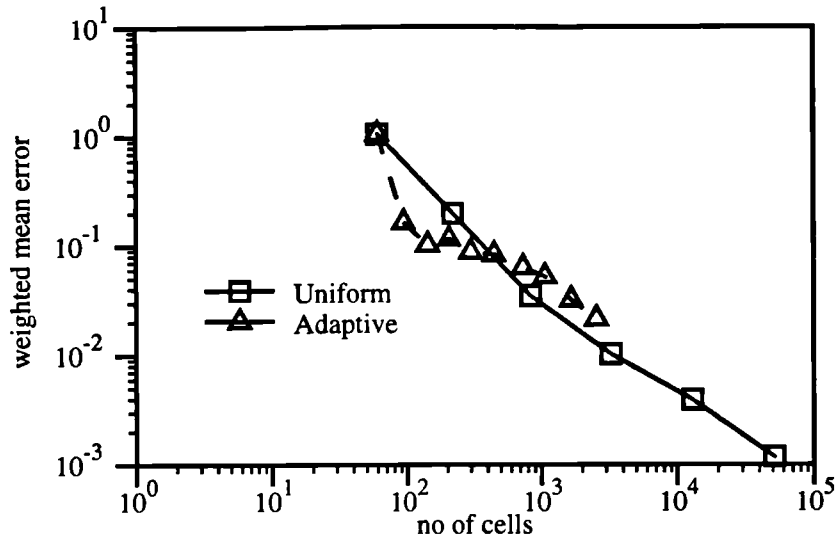


Figure 5.18: Uniform and adaptive refinement: scaling of the volume-weighted mean error.

appropriate way, the unweighted mean error will be used in the rest of this study. The sampling effect should be kept in mind when comparing the reduction of the mean error for uniform and adaptive refinement.

The maximum error, Fig. 5.17, does not suffer from the sampling problem. Its reduction in adaptive refinement is clearly much faster than in the case of uniform refinement, indicating an improvement in the overall accuracy for the same number of cells. The reduction of the maximum error still does not give the full picture of the solution quality. Adaptively refined meshes resolve the regions of high gradients considerably better than uniform meshes with the same number of cells, thus giving a better picture of the real solution.

In order to provide a more objective comparison of the uniform and adaptive refinement, we shall return to the concept of optimal meshes. According to the definition from Section 4.6, an optimal mesh implies a uniform distribution of the discretisation error. It is therefore possible to measure the mesh “optimality” as the

ratio of the mean and maximum error:

$$\eta = \frac{e_{mean}}{e_{max}}. \quad (5.15)$$

For $\eta = 1$, the error is uniformly distributed through the domain, as $e_{mean} = e_{max}$. In the case of inappropriate resolution in a certain part of the mesh, the maximum error will be larger than e_{mean} , resulting in a low η . Ideally, one would like to keep η as close to unity as possible. For low η , a local refinement procedure is expected to considerably improve the error distribution, causing a rapid decrease in the maximum error. It should, however, be noted that η bears no information about the error level. The variation of the mesh optimality index η for uniform and adaptive refinement with the number of cells is shown in Fig. 5.19. In the case

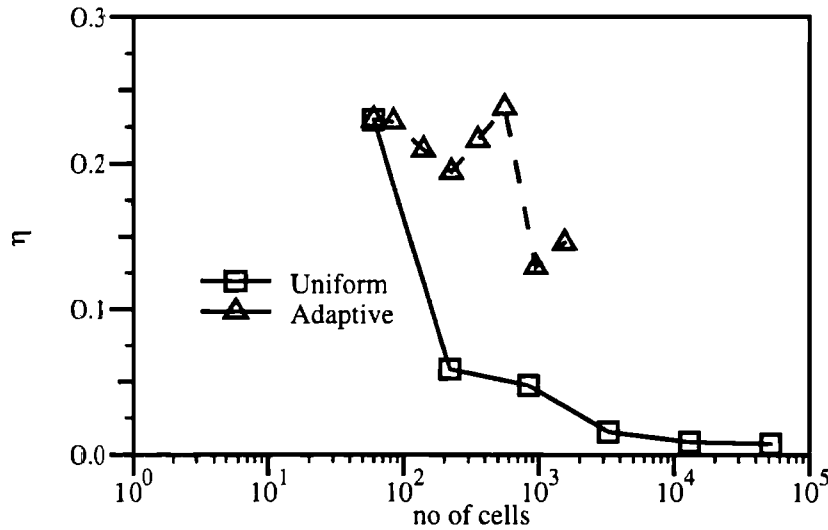


Figure 5.19: Optimality index for uniform and adaptive refinement.

of uniform refinement, new computational points are added uniformly through the domain, which is not a very efficient way of reducing the peak error. The mean error is being reduced faster, creating a small region of high error and decreasing the optimality index. The adaptive procedure, on the other hand, concentrates the refinement on the region of high error, thus improving the optimality. Both optimality indices are still far from unity, which is typical for the problems with a localised high error.

The “optimality” measure, Eqn. (5.15) should not be confused with the mesh quality. The quality of the mesh is judged by the level of mesh-induced errors caused by the cell distortion, non-orthogonality and mesh skewness. Although the two issues are clearly connected, the difference can be drawn in the following way: **mesh quality** is a property of the mesh, specifying the amount of mesh-induced errors expected from a certain mesh. **Mesh optimality**, on the other hand, depends on the interaction between the mesh and the particular problem that is being solved. From the optimality point of view, the main cause of low η is inappropriate mesh spacing in relation to the solution gradients, rather than the mesh-induced errors.

In order to present a more realistic picture of the influence of mesh changes to the accuracy, the graphs showing the cell centre values in comparison with the exact solution will be given. For this purpose, a cut through the domain, 0.2 m downstream from the inlet boundary and normal to the flow is made. Figs. 5.20, 5.21 and 5.22 show the exact and numerical solution on the initial mesh and after the first two levels of refinement (the meshes in question can be seen in Figs. 5.4, 5.5 and 5.6). The principle of adaptivity and the improvement in accuracy can now be clearly seen: the computational points are added only when they are needed. The solution after only two levels of adaptivity (Fig. 5.22) is accurate enough for all practical purposes. Further downstream, the smoothly-changing solution does not require the same number of computational points for the same level of accuracy. This is demonstrated in Fig. 5.23, showing the cut through the domain 3 m downstream from the inlet. The adaptively refined mesh resolves the profile with only 6 points, but this is completely appropriate in comparison with the exact solution. The same trend can be seen after ten levels of refinement, Figs. 5.24 and 5.25. The number of computational points is large where the solution varies rapidly and depends on the local curvature. Further downstream, the same level of accuracy can be obtained with a much smaller number of cells.

As described in Section 5.2, the selection of cells for refinement is based on the estimate of the local error. It could therefore be expected that the error reduction rate depends not only on the applied refinement technique, but also on the accuracy

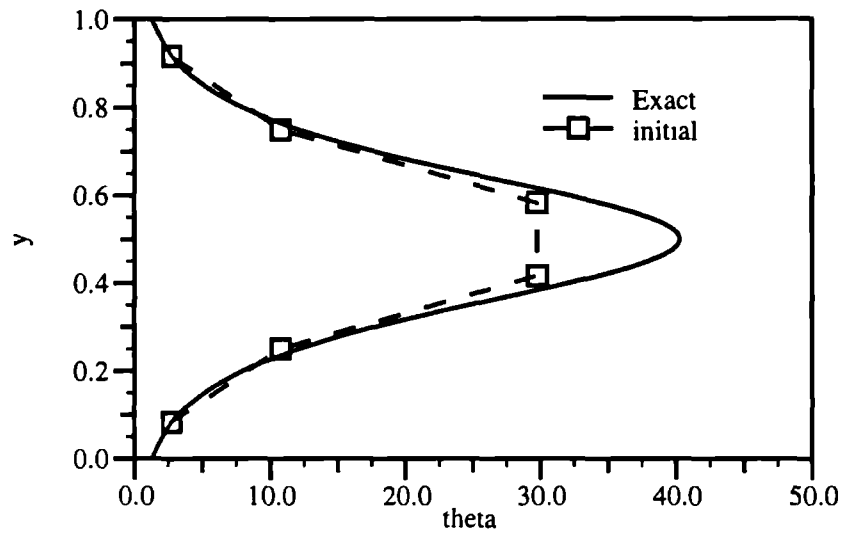


Figure 5.20: Refinement/unrefinement: solution at $x = 0.2\text{ m}$ on the initial mesh.

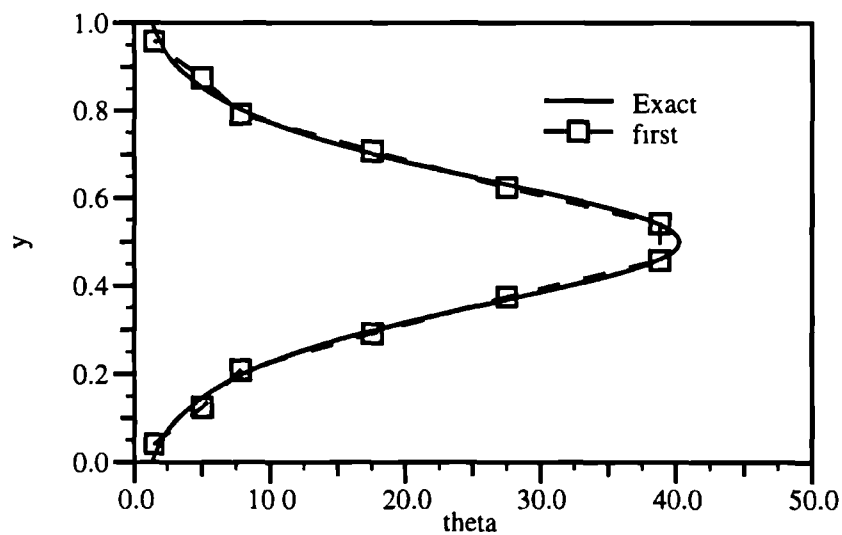


Figure 5.21: Refinement unrefinement. solution at $x = 0.2\text{ m}$ after the first level of refinement.

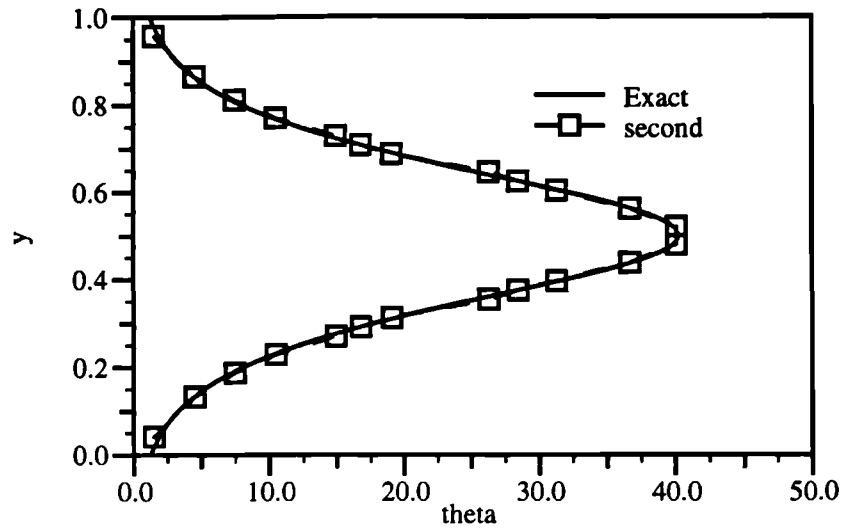


Figure 5.22: Refinement/unrefinement: solution at $x = 0.2m$ after the second level of refinement.

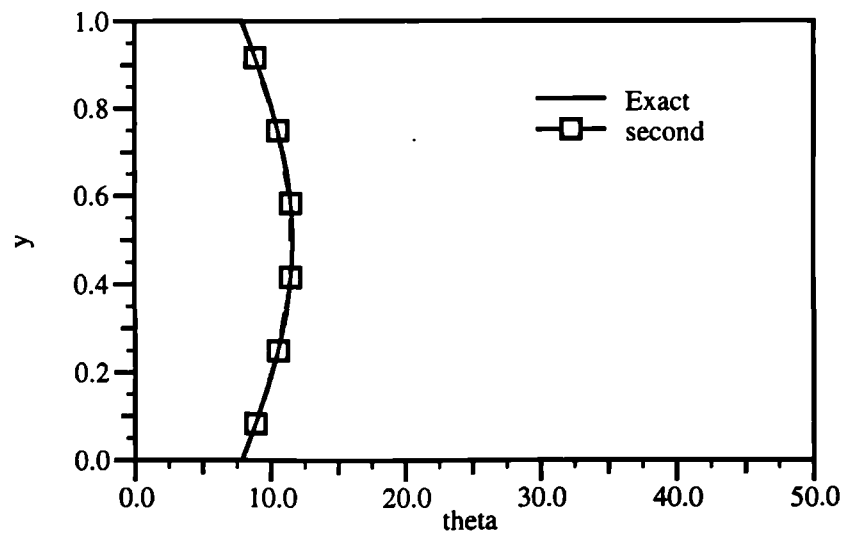


Figure 5.23: Refinement/unrefinement: solution at $x = 3m$ after the second level of refinement.

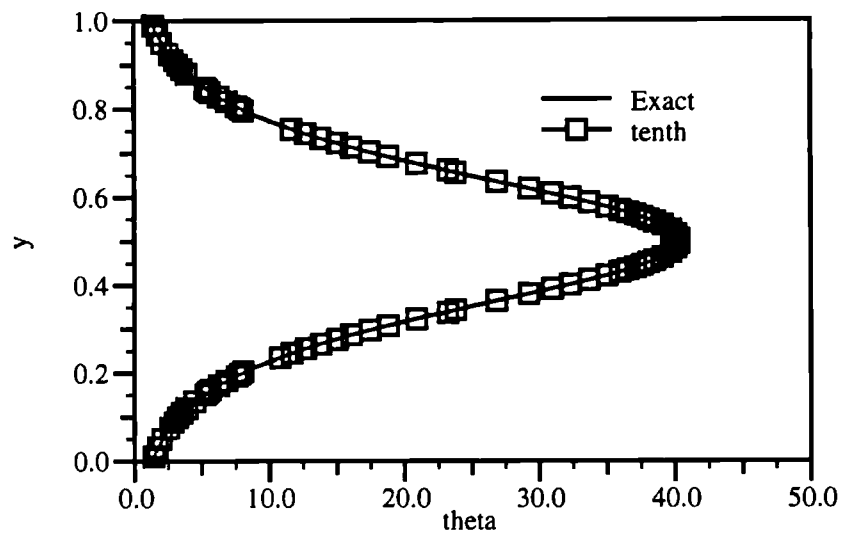


Figure 5.24: Refinement unrefinement: solution at $x = 0.2m$ after the tenth level of refinement.

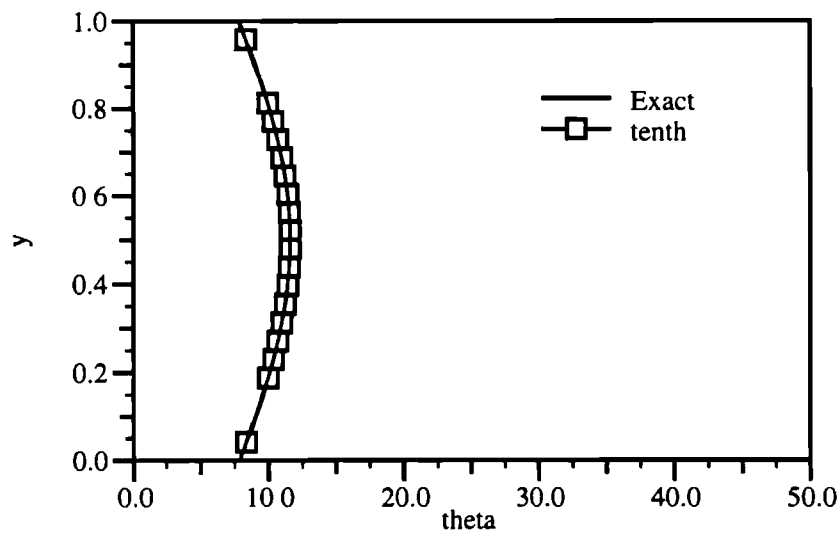


Figure 5.25: Refinement unrefinement: solution at $x = 3m$ after the tenth level of refinement.

of the error estimate. In order to examine the importance of this interaction, the adaptive procedure has been repeated, basing the choice of refinement regions on the exact error and each of the error estimates described in Chapter 4. Scaling of the mean and maximum of the exact error with the number of computational points for different error estimates is shown in Figs. 5.26 and 5.27. Several interesting conclusions can be drawn:

- The scatter of the points in Fig. 5.26 is relatively small, indicating similar performance of all error estimates.
- The adaptive refinement technique moves the error from one region of the domain to the other, sometimes creating meshes with high mesh-induced errors. This can be seen in the behaviour of the maximum error, Fig. 5.27. As the error estimates are typically sensitive to the mesh quality, the mesh-induced errors are removed in consequent mesh refinements.
- The adaptive procedure based on the exact error exhibits the slowest reduction of both mean and maximum error. Although this behaviour is by no means obvious, its cause can be explained from the nature of the exact error. Section 4.8 distinguishes two parts of the exact error: the locally induced error and the transported error. An adaptive procedure based on the exact error cannot distinguish between the two and, as a consequence, adds unnecessary resolution in the regions where only the transported error exists. Error estimates proposed in the present study neglect the mechanism of error transport from the region of high error sources to the rest of the domain, resulting in a more economical error removal.
- In the final stages of refinement, the maximum error reaches a plateau (see Fig. 5.27) and the amount of refinement needed to further improve the accuracy becomes very large. At this stage, embedded refinement is spread through the whole of the domain, creating strongly linked refinement patterns. The “1-irregularity” condition introduces numerable changes in the mesh for every prescribed cell split. The mesh quality (in terms of average skewness and

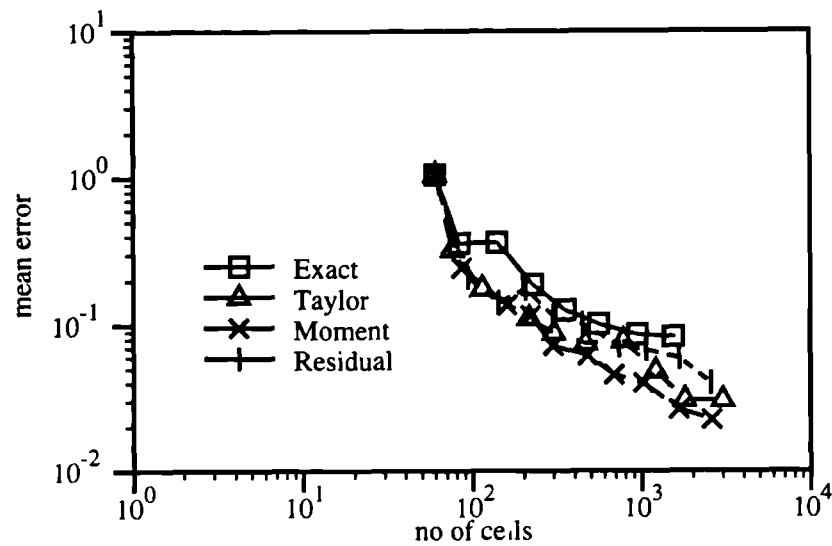


Figure 5.26: Adaptive refinement: scaling of the mean error for different error estimates and the exact error.

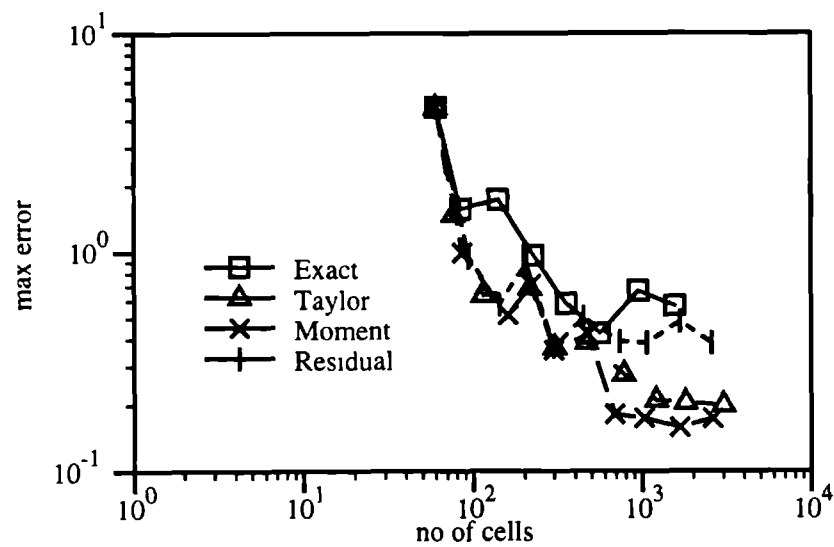


Figure 5.27: Adaptive refinement: scaling of the maximum error for different error estimates and the exact error r

non-orthogonality) is severely reduced. The bulk of the error should now be attributed to the mesh quality rather than insufficient resolution in the regions of high gradients. Since the objective of adaptivity is to produce accurate solutions with only a slight increase in computational cost, further refinement is not considered to be efficient. If the solution accuracy is still not appropriate, the adaptive procedure should be repeated from a finer initial mesh. As a guideline, in this particular case the plateau was reached when the initial maximum error was reduced by a factor of 20. This is a property of the particular mesh adaptation algorithm rather than the adaptive approach itself. If it were possible to create an adaptive procedure that does not impair the mesh quality, this behaviour could be avoided altogether.

The second approach to the adaptive calculations starts from a relatively fine initial mesh and use both refinement and unrefinement to improve the mesh quality. In order to examine the performance of the unrefinement procedure, both refinement-only and refinement/unrefinement calculations will be performed. The initial mesh consists of 80×40 CV (Fig. 5.28), which is considered to be too fine in the smooth parts of the solution and too coarse close to the singularity. The refinement/unrefinement parameters are set to:

$$\begin{aligned}\lambda_{ref} &= 1.5, \\ \lambda_{unref} &= 0.5, \\ \xi &= 0.6.\end{aligned}$$

Unlike refinement, the unrefinement algorithm adopted in this study does not preserve the symmetry of the mesh for symmetric problems. The symmetry would require some kind of global optimisation: the unrefinement procedure would have to recognise the plane of symmetry not imposed from the outside. The adopted algorithm selects the “cell pairs” on a cell-by-cell basis, with the final result depending, among other things, on the order of cells in the cell list.

Although aesthetically pleasing, the symmetry is not essential for the accuracy of the solution. If the mesh quality is deteriorated as a result of a “non-optimal”

selection of cell pairs, consequent refinement levels are expected to remove the error.

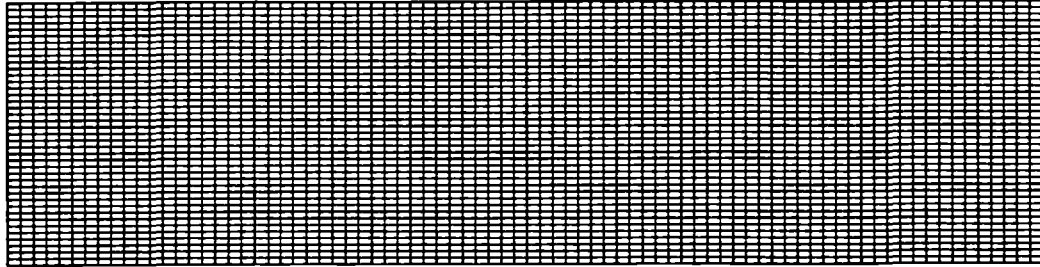


Figure 5.28: Initial mesh for refinement/unrefinement.

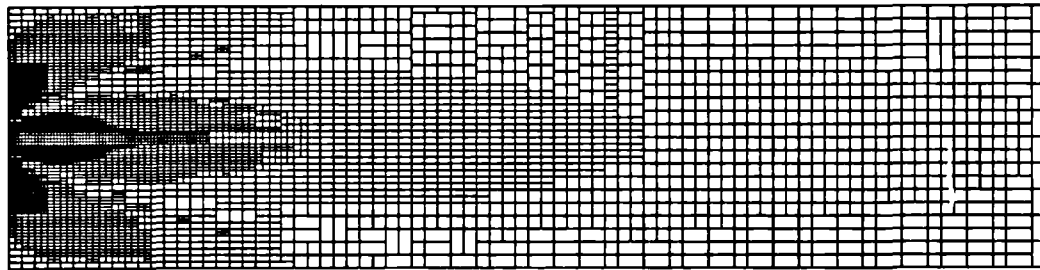


Figure 5.29: Second level of refinement/unrefinement.

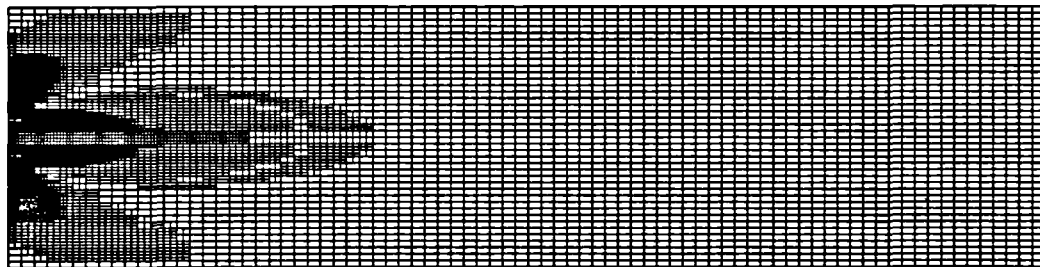


Figure 5.30: Second level of refinement-only.

An example (asymmetric) mesh resulting from refinement/unrefinement is shown in Fig. 5.29, with the equivalent refinement-only mesh in Fig. 5.30. In the region of smooth gradients far downstream from the point source, the estimated error is much lower than average. The cells in that region have been merged to create larger cells, resulting in lower local accuracy. Some parts of the mesh have been left unchanged, as the local accuracy corresponds to the average accuracy of the solution. In the regions of high error close to the point source, the mesh resolution

has been improved.

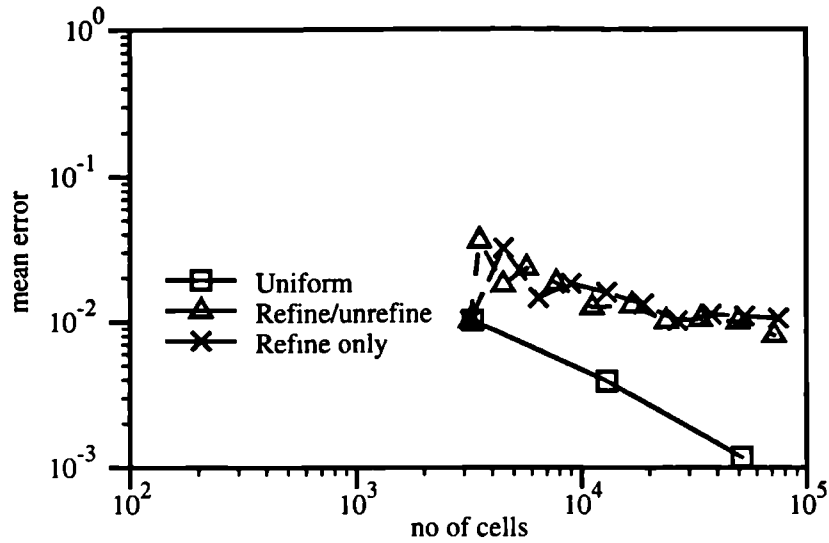


Figure 5.31: Refinement/unrefinement: scaling of the mean error.

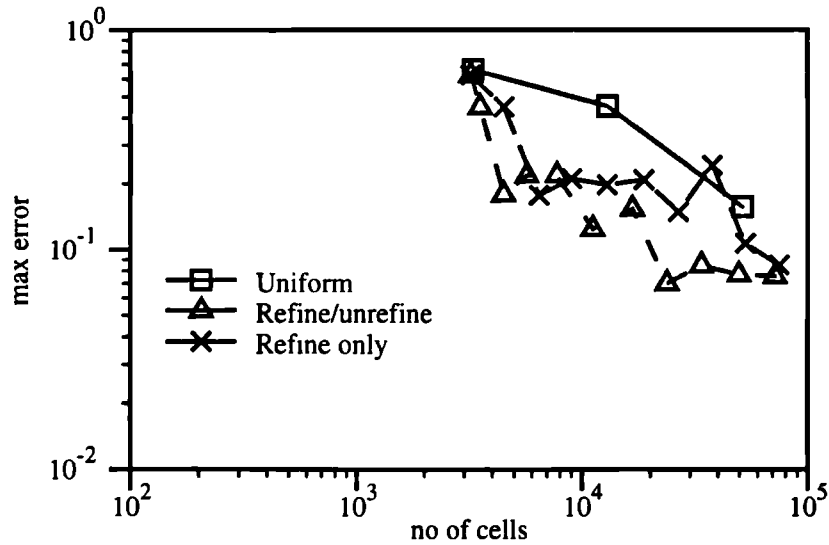


Figure 5.32: Refinement/unrefinement: scaling of the maximum error.

A comparison of the mean and maximum error scaling for adaptive refinement based on the Residual Error estimate is shown in Figs. 5.31 and 5.32. The sampling problem can now be seen even more clearly. The scaling of the mean error suggests that uniform refinement reduces the error more efficiently than both refinement-only and refinement unrefinement. The reduction of the maximum error, Fig. 5.32

is again faster for the adaptive procedure than in the case of uniform refinement. The plateau in the maximum error is reached when the initial peak error is reduced by a factor of 10. It should, however, be noted that the error level on the initial mesh is of the order of 1 %, which is lower than the typical accuracy needed for engineering purposes.

A closer look into Figs. 5.31 and 5.32 reveals the correspondence between the refinement only and refinement/unrefinement calculations. The peak error is in both cases located in the refined region and its level is approximately the same. The unrefinement procedure merges the cells in the region of low local error, thus moving both graphs towards lower numbers of computational points. Refinement/unrefinement therefore reaches the maximum error plateau with the smaller number of cells than refinement-only. In terms of the mean error, the difference between the two adaptive procedures is marginal.

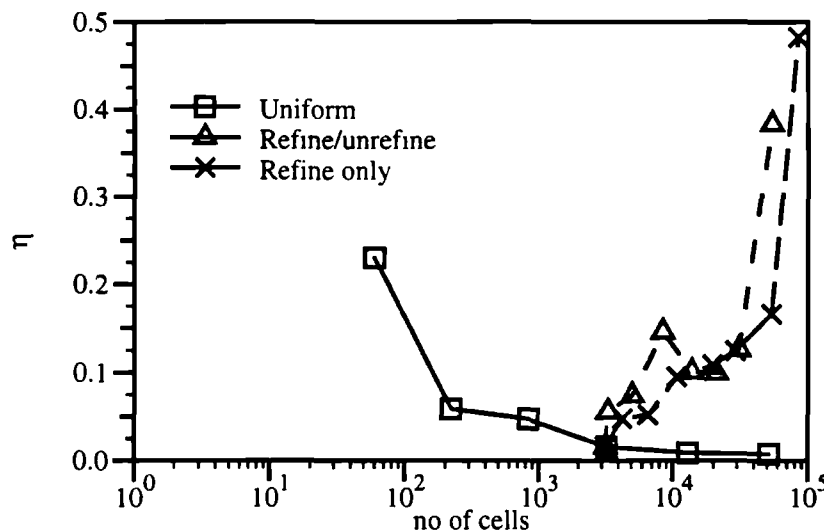


Figure 5.33: Optimality index for uniform refinement, adaptive refinement-only and adaptive refinement unrefinement.

The variation of η for refinement-only and refinement/unrefinement is shown in Fig. 5.33. Both adaptive procedures increase the optimality index very rapidly. The initial increase is larger for refinement unrefinement, as the removal of unnecessary cells influences the mean error. It should also be noted that η for the final adaptively refined mesh is much higher than for any uniform mesh in spite of the additional

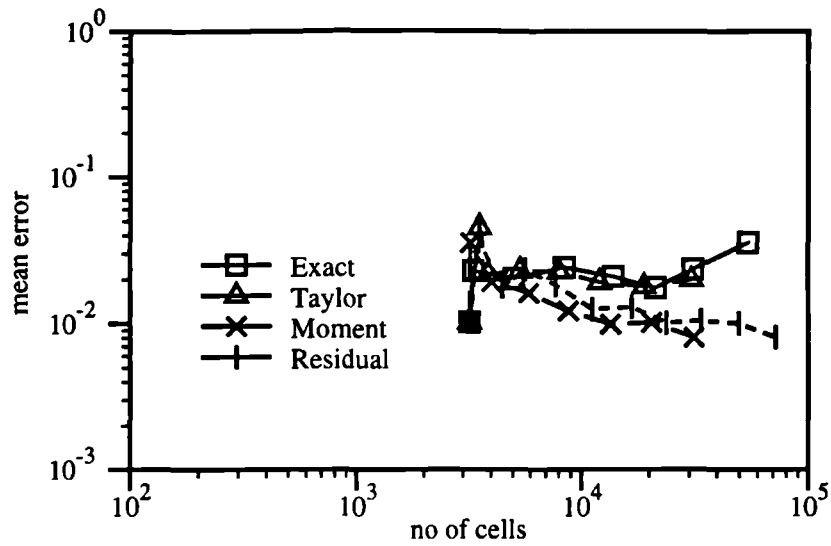


Figure 5.34: Refinement/unrefinement: scaling of the mean error for different error estimates.

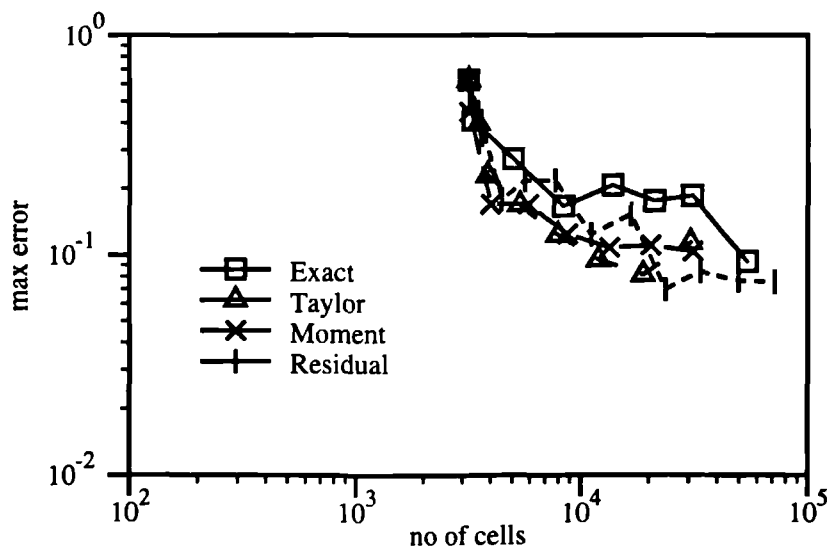


Figure 5.35: Refinement/unrefinement: scaling of the maximum error for different error estimates.

resolution added in the region of high error.

Let us finally examine the influence of error estimation on the error reduction rate in the case of refinement/unrefinement. For that purpose, the adaptive calculation has been repeated with the use of the proposed error estimates and the exact error. The scaling of the mean and maximum error with the number of cells, for different methods of estimation are shown in Figs. 5.34 and 5.35. The error reduction follows a similar path for all error estimates, with the one based on the exact error again showing the slowest drop. It can be therefore concluded that all error estimates accurately highlight the error sources and the differences in the estimated error distribution do not have a major influence on the adaptive process.

5.6 Closure

This Chapter presented some details of the adaptive local mesh refinement and unrefinement procedure used in this study. The mesh adaptation algorithm is based on the results of the error estimation procedure described in Chapter 4. Firstly, the algorithms for an error-driven adaptive mesh generation and mesh refinement procedure have been given. They consist of several parts: the analysis of the error distribution, selection of regions for refinement and unrefinement, changing the actual computational mesh, and finally, mapping the solution from one mesh to the other.

In order to overcome the problems connected with the issue of automatic mesh generation, the selected procedure aims to change the existing computational mesh in an optimal way. Smooth mesh grading is preserved through the concept of “1-irregularity” of the mesh.

The mesh refinement procedure is controlled by three parameters: λ_{ref} and λ_{unref} , controlling the grading between the coarse and fine regions of the mesh, and ξ determining the desired level of mesh alignment.

When the new mesh is created, an interpolate of the solution from the previous mesh is used as an initial guess. Consistent with the discretisation practice, it is

assumed that the variation of the function over the control volume is linear. Some attention needs to be given to the issue of the transfer of fluxes between two meshes. A procedure that produces a set of conservative fluxes irrespective of the topological interaction between the meshes has been described in Section 5.4. This completes the adaptive local mesh refinement and unrefinement algorithm. An example of its performance on a test case with an analytical solution has been given in Section 5.5.

Chapter 6

Case Studies

6.1 Introduction

In this Chapter, the efficiency of the proposed error-driven adaptive mesh refinement algorithm will be examined on several test cases. Mesh adaptation can be performed accurately only if it is based on an accurate estimate of the error. Section 4.8 presented a comparison of the exact and estimated errors for several test cases with analytical solutions. In real flow situations, this is not possible as the governing equations cannot be solved analytically. It is, however, still possible to solve the flow problem on a very fine mesh and consider the solution to be accurate enough to provide a good estimate of the exact error. In computational terms, this is extremely expensive and practical only for two-dimensional laminar test cases.

The performance of adaptive refinement algorithms is most effective on problems with discontinuous solutions. High resolution is needed only along the line of the discontinuity in 2-D, or along the discontinuity surface in 3-D. From the point of mesh resolution, the adaptive refinement algorithm effectively reduces the dimensionality of the problem by one, resulting in significant reductions in the number of cells required to obtain a solution of a certain accuracy. A case of this kind, involving two-dimensional supersonic inviscid flow over a forward-facing step will be presented in Section 6.2. The performance of both adaptive mesh refinement and refinement unrefinement procedures will be examined. In this situation, it is easy

to judge the accuracy of error estimates in spite of the fact that the exact solution is not available, as the error is represented by the finite thickness of the shocks. The mesh refinement algorithm is controlled by a pressure-based error indicator, which is consequently compared with the proposed error estimates. This error indicator is tailored for discontinuous flows and its known properties are used to judge the quality of the new error estimates.

The second test case is selected to illustrate the performance of the algorithm on steady incompressible recirculating flows, in both laminar and turbulent regimes. Unlike the supersonic test case, the incompressible flow over a hill produces a smooth solution, posing more serious demands on both error estimation and adaptive refinement. For the laminar case the “exact” numerical solution on a very fine mesh will be used to compare the error estimates with the exact error. In the next phase, turbulent flow will be examined, with two different wall treatments. The wall function approach compensates for the presence of the wall through certain modifications in the shear stress and turbulence variables in the near-wall cell. The accuracy of the solution therefore depends on the interaction of the size of the near-wall cell and the wall model, which should be taken into account both in error estimation and adaptive refinement. Low-Reynolds-number models, on the other hand, integrate the equations all the way to the wall and therefore do not require any special treatment, either in error estimation or in mesh refinement. The rapid variation in all fields close to the wall requires high mesh resolution and is likely to cause high local errors. In the case of turbulent calculations, the error from all the solution variables should be taken into account when the decision on mesh refinement is made. Ideally, the influence of the error in each of the variables on the local solution error should be taken into account through some form of error weighting. This would, however, require either an extensive parametric study for each point in the domain, or a good understanding of the (non-linear) inter-equation coupling. Considering the high cost of such an analysis and the limited gain in terms of the computational effort, all the errors will be treated as equally important.

Potential gains in the performance for adaptive algorithms on 3-D problems are

even larger than in two dimensions. If the mesh spacing is halved in every direction, the total number of cells in 3-D increases by a factor of 8, whereas in 2-D the number of cells quadruples. The turbulent flow over a swept backward-facing step, presented in Section 6.4 has been selected to illustrate the mesh adaptation in three dimensions.

Finally, in Section 6.5, a transient test case is examined. The transient error estimate can be used in several ways. Firstly, it is possible to examine the error balance in order to select the optimal time-step size for a given mesh. The balance depends on the interaction of the spatial and temporal error in different regions of the domain and the number of time-steps necessary to resolve the temporal evolution of the flow. In the case of vortex shedding behind a cylinder presented in this Section, the solution varies periodically in time. If the periodicity is regular, it is possible to follow the solution during one cycle and mark all the cells in which the error reaches some pre-determined threshold at any time. This information is then used to refine the mesh, reducing the spatial component of the error. The advantage of this approach is that the mesh does not change in time, reducing the overall cost of the adaptive calculation.

6.2 Inviscid Supersonic Flow Over a Forward-Facing Step

This two-dimensional test problem was introduced by Emery [42]. The domain is filled with an ideal gas with $\gamma = 1.4$. The inlet conditions are set to:

$$U = 1 \text{ m/s}$$

$$\rho = 1.4 \text{ kg/m}^3$$

$$P = 1 \text{ kg/ms}^2$$

which corresponds to the Mach number of 3. At the outlet boundary, a zero gradient condition is specified on all variables. Along the walls of the tunnel, the reflecting boundary condition is applied. The corner of the step is the centre of a rarefaction

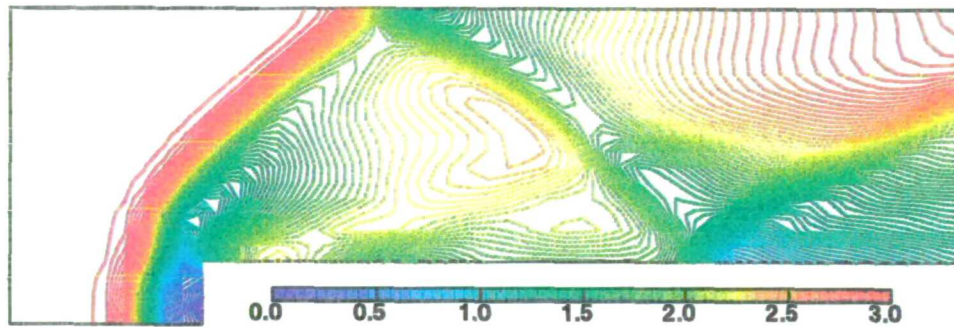


Figure 6.1: Supersonic flow over a forward-facing step: Mach number distribution, uniform mesh, 840 CV.

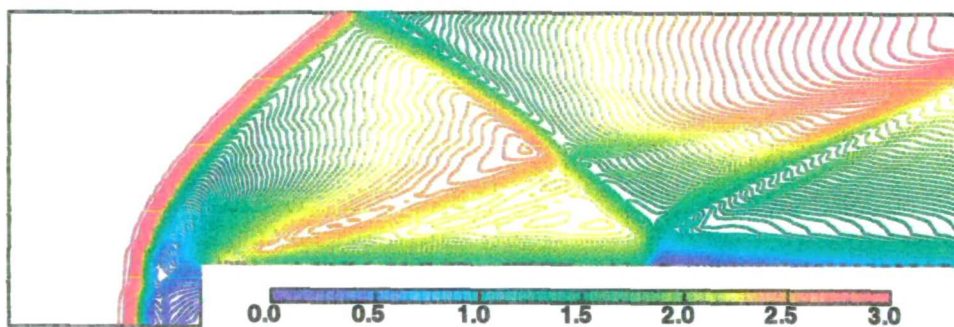


Figure 6.2: Supersonic flow over a forward-facing step: Mach number distribution, uniform mesh, 5250 CV.

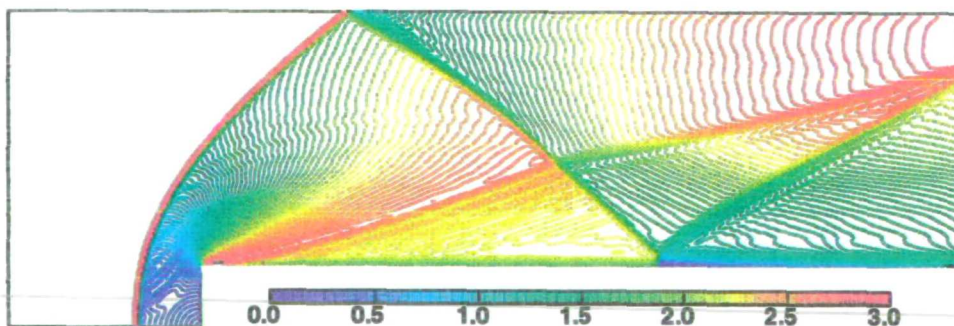


Figure 6.3: Supersonic flow over a forward-facing step: Mach number distribution, uniform mesh, 53000 CV.

fan and hence a singular point of the flow. This singularity has not been treated in any special way and the local error causes a boundary layer of about one cell thickness just above the step (Woodward and Colella [149]). A pressure-based supersonic flow solver is used. In order to avoid spurious oscillations in velocity and pressure, the Gamma differencing scheme is used on all equations.

In the first instance, the problem is solved on three uniform meshes, with 840, 5250 and 53000 CV-s, giving the Mach number distribution presented in Figs. 6.1, 6.2 and 6.3, respectively. An interesting feature is the over-expansion region at the corner, followed by a weak shock originating just downstream of the edge. The nature of the over-expansion and its interaction with the corner singularity has been briefly discussed by Woodward and Colella [149]¹. From the point of view of adaptive refinement, the presence of both weak and strong shocks complicates the situation considerably, particularly because the weak shock is not picked up in the coarse mesh (see Fig. 6.1).

In the first instance, the following pressure based error indicator will be used:

$$\Upsilon = \frac{|\nabla p|}{h^2} \quad (6.1)$$

The local cell size h is calculated from Eqn. (4.12). This error indicator measures the pressure jump over a cell, giving a good indication of the shock resolution in comparison with other flow features. This error indicator will be consequently used to judge the performance of the error estimators described in Chapter 4.

Figs. 6.4, 6.5 and 6.6 show the distribution of Υ on three uniform meshes, corresponding to the Mach number distribution from Figs. 6.1, 6.2 and 6.3. The error indicator correctly highlights the main shock structure on all meshes. The weak shock originating from the corner is recognised only on the finest mesh.

The error indicator can now be compared with the estimated error distribution. Figs. 6.7, 6.8 and 6.9 show the estimated errors on the intermediate mesh (5250 CV-s) using the Direct Taylor Series, Moment and Residual Error estimates for the momentum equation. Unlike the error indicator, all three error estimates

¹Similar effects occur in wind tunnel experiments of this type [149].

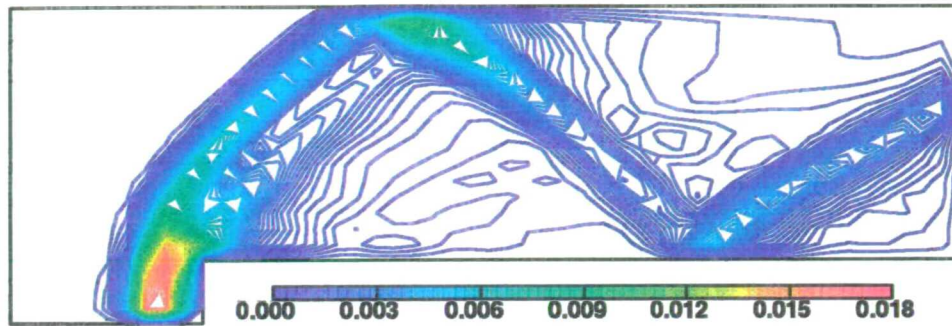


Figure 6.4: Supersonic flow over a forward-facing step: error indicator Υ , uniform mesh, 840 CV.

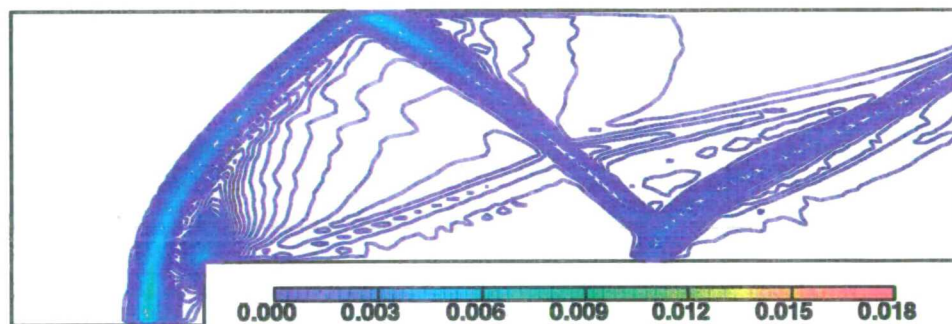


Figure 6.5: Supersonic flow over a forward-facing step: error indicator Υ , uniform mesh, 5250 CV.

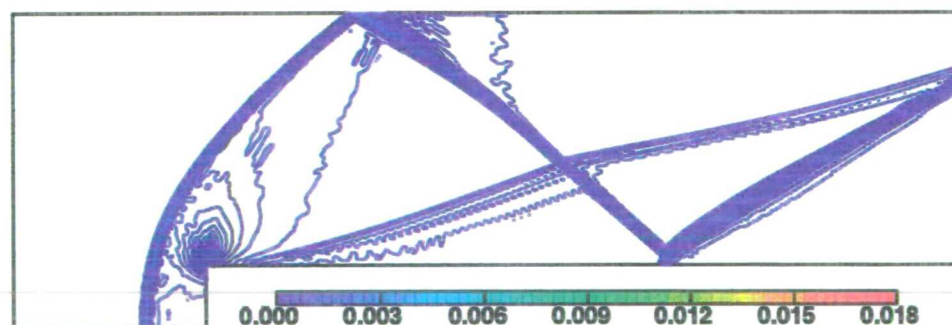


Figure 6.6: Supersonic flow over a forward-facing step: error indicator Υ , uniform mesh, 53000 CV.

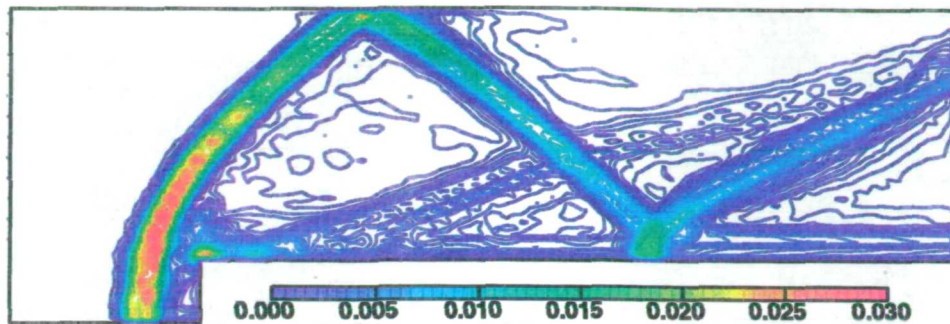


Figure 6.7: Supersonic flow over a forward-facing step: Direct Taylor Series Error estimate for the momentum, uniform mesh, 5250 CV.

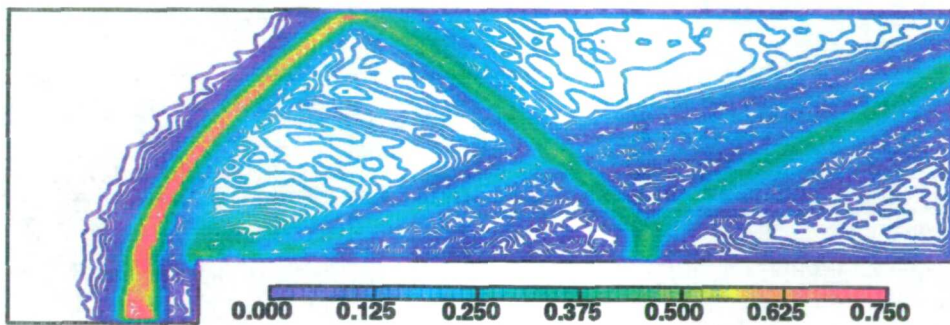


Figure 6.8: Supersonic flow over a forward-facing step: Moment Error estimate for the momentum, uniform mesh, 5250 CV.

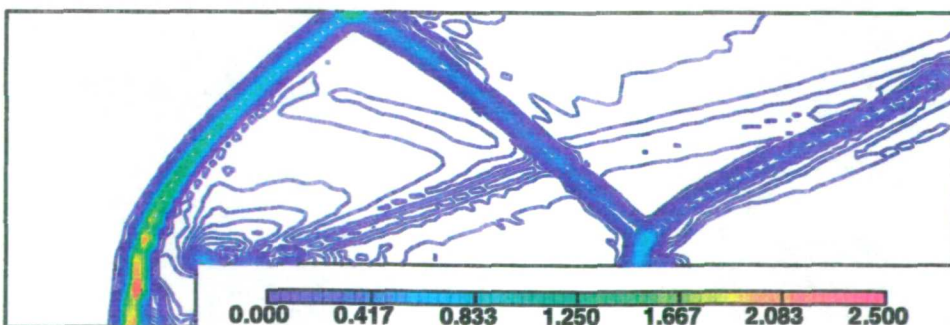


Figure 6.9: Supersonic flow over a forward-facing step: Residual Error estimate for the momentum, uniform mesh, 5250 CV.

successfully locate the weak shock at the corner of the step.

The adaptive strategies described in Chapter 5 will now be used: refinement-only from the coarse mesh (840 CV, Fig. 6.10), refinement-only from the intermediate mesh (5250 CV, Fig. 6.11) and refinement/unrefinement from the same initial mesh.

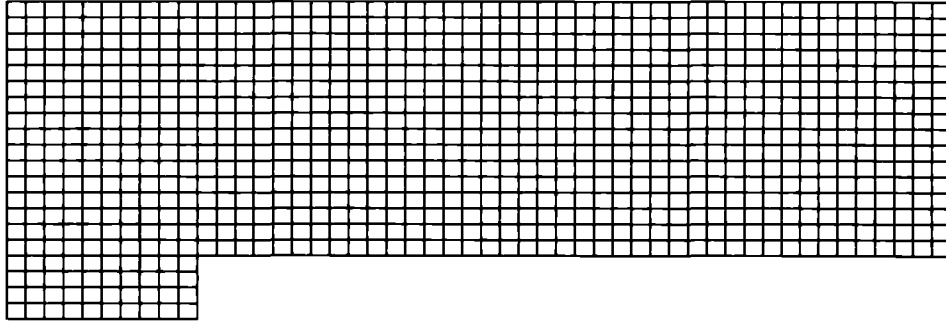


Figure 6.10: Supersonic flow over a forward-facing step: coarse mesh, 840 CV.

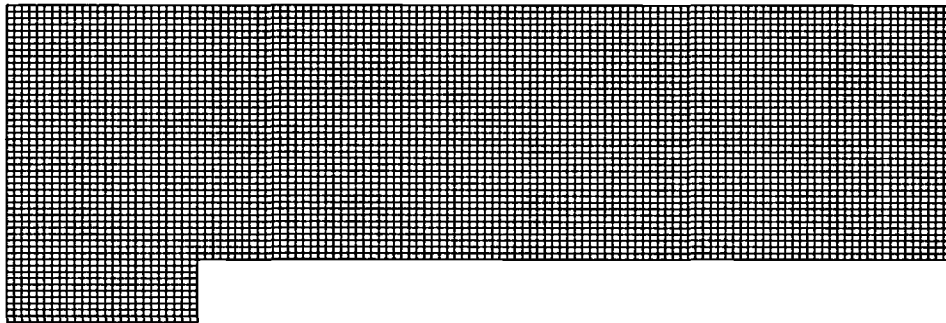


Figure 6.11: Supersonic flow over a forward-facing step: intermediate mesh, 5250 CV.

The refinement parameters are set to:

$$\begin{aligned}\lambda_{ref} &= 1.5 \\ \lambda_{ref} &= 0.5 \\ \xi &= 0.6\end{aligned}$$

In Figs. 6.12, 6.13 and 6.14, the changes in the mesh for the first three levels of refinement from the coarse initial mesh guided by Eqn. (6.1) are shown. Following the error indicator, additional cells are added around the main shock structure. However, the error indicator Υ does not recognise the weak shock since it does not

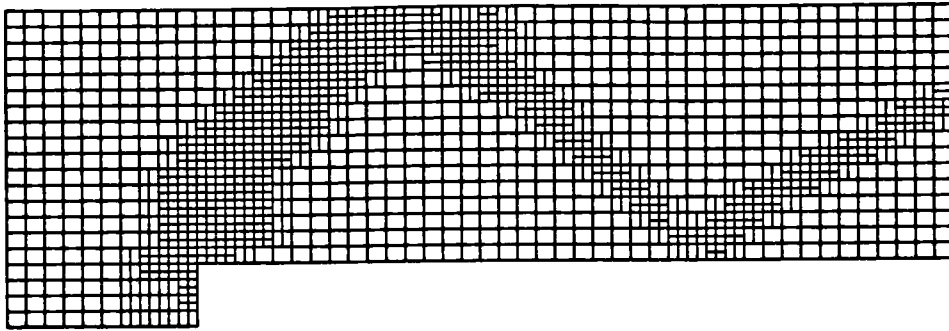


Figure 6.12: Supersonic flow over a forward-facing step: first level of adaptive refinement starting from the mesh in Fig. 6.10.

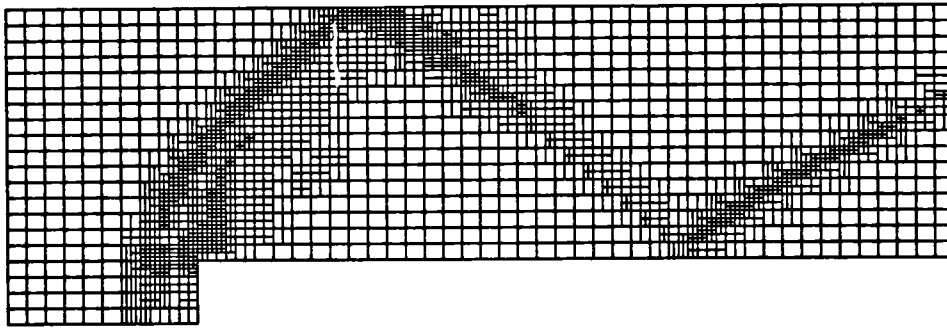


Figure 6.13: Supersonic flow over a forward-facing step: second level of adaptive refinement starting from the mesh in Fig. 6.10.

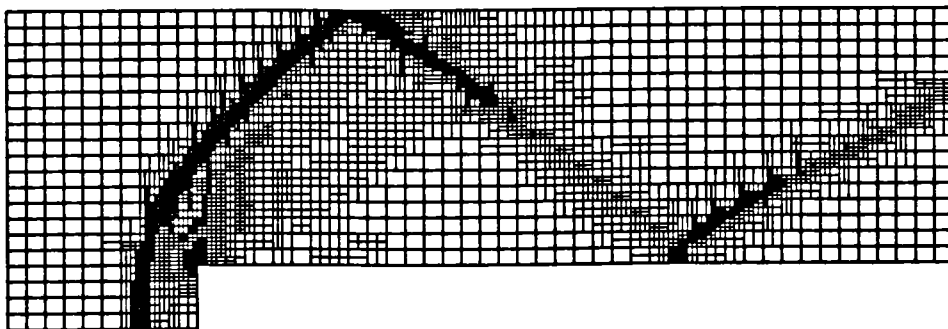


Figure 6.14: Supersonic flow over a forward-facing step: third level of adaptive refinement starting from the mesh in Fig. 6.10.

exist in the coarse mesh solution (Fig. 6.1) and as a consequence, no new cells are added in the region around it.

A detail of the refined mesh, showing the “1-irregularity” principle in action is shown on Fig. 6.15. The grading between the coarse and fine parts of the mesh is

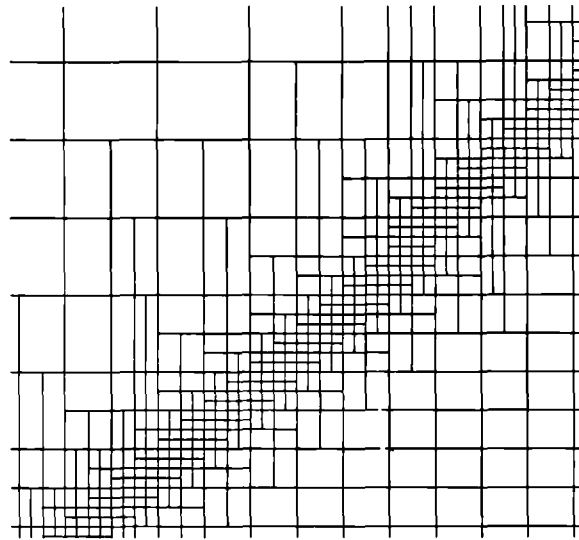


Figure 6.15: Detail of the adaptively refined mesh.

relatively smooth, preserving the quality of the refined mesh around the shock. If the “1-irregularity” were not obeyed, rapid grading would cause high mesh-induced errors, degrading the quality of the numerical solution.

The Mach number distribution after three levels of refinement is shown in Fig. 6.16 and the final solution, after five refinement levels in Fig. 6.17. The shock resolution on the adaptively refined mesh with 15220 CV-s, (Fig. 6.17) is clearly superior to the solution on a uniform mesh with 53000 CV-s (Fig. 6.3), demonstrating the potential of the adaptive algorithm. The quality of the solution, however, cannot be judged only by the resolution of strong shocks. The fine uniform mesh resolves the weak shock originating from the corner of the step reasonably well. Although the adaptive solution picks up the same feature, its resolution is not comparable with the resolution of the main shock structure. After three levels of refinement, Fig. 6.16, the weak shock is not picked up at all. This is clearly an undesirable side-effect of adaptive refinement. The key to the problem lies in the solution used to start the

adaptive procedure. On the coarse mesh, the weak shock in question is not picked up at all (see Fig. 6.1). The improvement of its resolution is possible only when the mesh refinement progresses far enough to detect the shock in the first place.

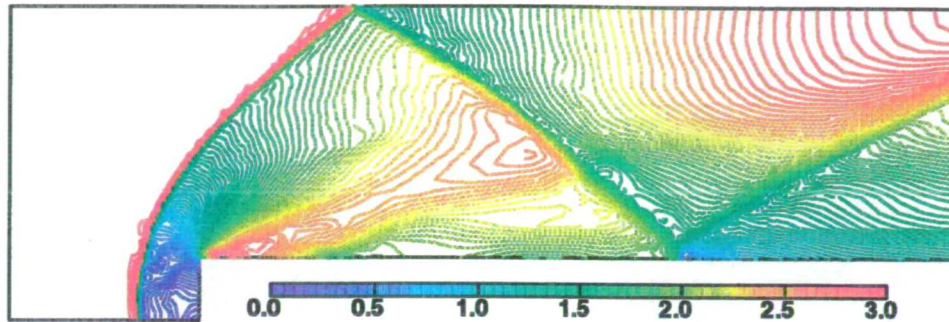


Figure 6.16: Supersonic flow over a forward-facing step: Mach number distribution on the adaptively refined mesh after three levels of refinement.

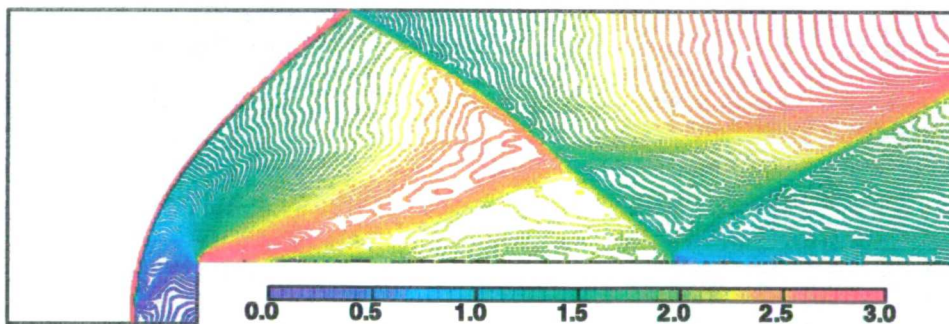


Figure 6.17: Supersonic flow over a forward-facing step: Mach number distribution on the adaptively refined mesh after five levels of refinement.

The preferred approach to the adaptive calculations would be to start from a mesh that is capable of picking up all the features of the flow, leading us to the second adaptive refinement strategy. The adaptive procedure starts from the intermediate mesh (5250 CV-s, Fig. 6.11) and in the first instance performs refinement only. The final mesh, after three levels of adaptive refinement, is shown in Fig. 6.18. This mesh is considerably too fine in the region close to the inlet boundary. In this part of the domain all the fields are constant, but the refinement-only procedure has no means of removing the unnecessary computational points. The resolution of the weak shock is now adequate, as it was possible to recognise it in the early stages of

refinement.

This test case illustrates a dilemma in the selection of the initial mesh for adaptive calculations. If it is too coarse, some of the important flow features may be left out of the initial stages of refinement, degrading the quality of the final solution. If, on the other hand, the initial mesh is too fine, a considerable number of computational points may end up in the region where they are not needed. This dilemma can be resolved by the use of a refinement/unrefinement procedure. The initial mesh can now be fine enough to resolve all the features of the flow, and the computational points can be removed if they are not needed.

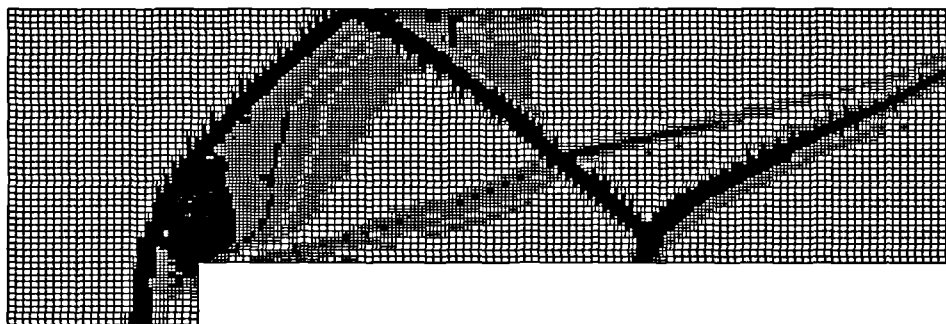


Figure 6.18: Supersonic flow over a forward-facing step: third level of adaptive refinement starting from the mesh in Fig. 6.11.

The first three levels of refinement/unrefinement starting from the same initial mesh are shown in Figs. 6.19, 6.20 and 6.21. The merging of cells in the region close to the inlet can be clearly seen, as the local error in that region is effectively zero. It can also be seen that in spite of the inter-locking mentioned in Section 5.5, the quality of the refined mesh is still adequate, as is the resolution of the weak shock. It is interesting to follow the progress of mesh refinement in the region surrounding it. The first level of refinement/unrefinement, Fig. 6.19, leaves the mesh resolving the weak shock intact, while the region around it is coarsened. The second and third level add more computational points around the shock, specially where it interacts with the strong reflected shock.

The comparison of the Mach number distribution for refinement only, Fig. 6.22, and refinement/unrefinement, Fig. 6.23, after three levels of refinement reveals no

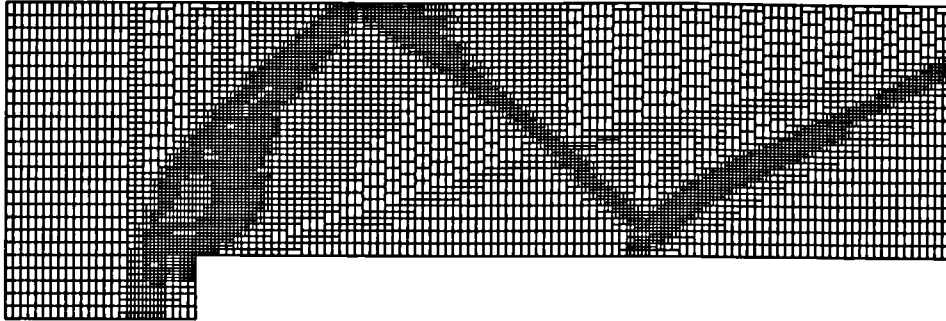


Figure 6.19: Supersonic flow over a forward-facing step: first level of adaptive refinement/unrefinement starting from the mesh in Fig. 6.11.

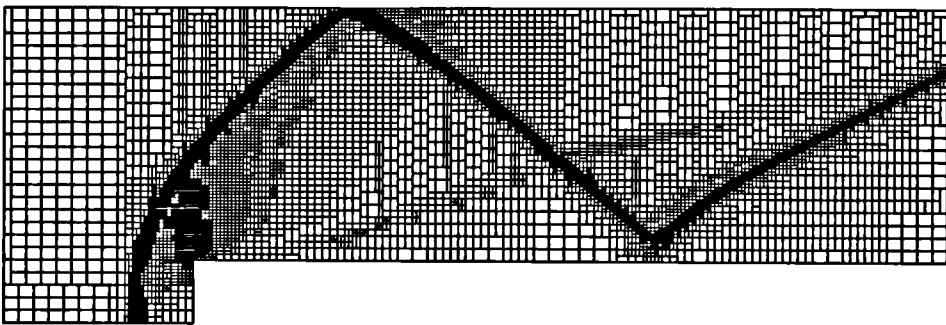


Figure 6.20: Supersonic flow over a forward-facing step: second level of adaptive refinement/unrefinement starting from the mesh in Fig. 6.11.

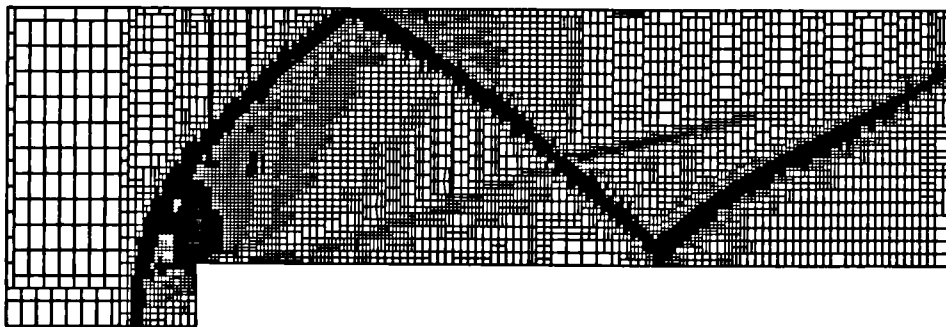


Figure 6.21: Supersonic flow over a forward-facing step: third level of adaptive refinement/unrefinement starting from the mesh in Fig. 6.11.

considerable differences.

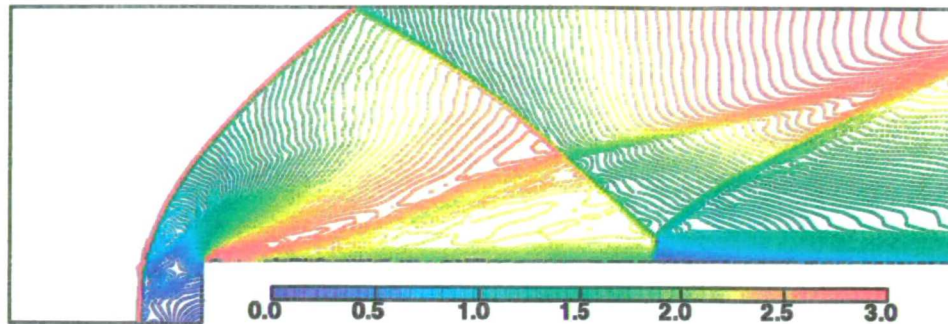


Figure 6.22: Supersonic flow over a forward-facing step: Mach number distribution for the mesh in Fig. 6.18.

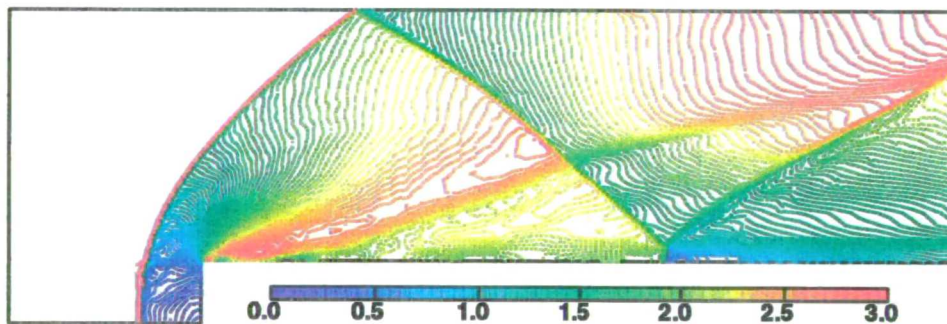


Figure 6.23: Supersonic flow over a forward-facing step: Mach number distribution for the mesh in Fig. 6.21.

The objective of the adaptive procedure is to produce the accurate solution with the minimum number of computational points. The performance of the algorithm can therefore be measured in terms of the number of cells used *vs.* the number of cells in the uniform mesh giving the equivalent resolution. Table 6.1 shows the increase in the number of cells in four levels of adaptive refinement starting from the coarse initial mesh and the number of cells for the uniform mesh with equivalent resolution. The percentage ratio in the second column of Table 6.1 gives the number of computational points in the adaptively refined mesh as a fraction of the number of cells in the uniform mesh. This can be used to estimate the reduction in the computational effort needed to obtain a solution of a certain accuracy between adaptive and uniform mesh refinement. The resolution of the initial mesh is inadequate in

Adaptive refinement	% ratio	Uniform
840	100 %	840
1440	42.8 %	3360
2396	17.8 %	13440
4160	7.73 %	53760
7867	3.66 %	215040
15220	1.77 %	860160

Table 6.1: Supersonic flow over a forward-facing step: number of cells for adaptive and uniform refinement starting from the coarse mesh.

most of the domain and the early stages of refinement introduce a considerable number of cells. As the refinement progresses, the benefit of the adaptive procedure becomes more and more apparent.

Refinement only	% ratio	Refinement and unrefinement	% ratio	Uniform
5250	100 %	5250	100 %	5250
7690	36.6 %	6200	29.5 %	21000
11748	14.0 %	8796	10.5 %	84000
18903	5.62 %	14738	4.38 %	336000
33219	2.47 %	27091	2.01 %	1344000
60493	1.12 %	54222	1.01 %	5376000

Table 6.2: Supersonic flow over a forward-facing step: number of cells for refinement-only and refinement/unrefinement starting from the intermediate mesh.

Table 6.2 shows the respective number of cells for refinement-only and refinement/unrefinement starting from the intermediate mesh. The refinement-only procedure behaves in approximately the same way as in Table 6.1, although the initial benefit is somewhat larger. Unrefinement brings substantial benefits only in the first

three steps. As the refinement progresses, the difference between the two meshes decreases, as more of the cells are placed where they are actually needed. The benefit from unrefinement is also potentially larger if the starting mesh is finer. This will generally be the case: when the shape of the solution is not known in advance, the analyst will tend to create a mesh which is fine everywhere, knowing that the adaptive procedure is capable of removing the excess cells where they are not needed.

Attention will now be turned to error estimation for supersonic flows. From the numerical accuracy point of view, supersonic flows are singular and the shock resolution is inadequate irrespective of the cell size. On uniform meshes the maximum error, Fig. 6.25, is therefore independent of mesh refinement. The mean error, Fig. 6.24, decreases as the mesh gets finer, partly because of the better resolution of the smooth profiles and partly because the number of cells away from the shock increases relative to the total number of cells.

The error scaling in the case of adaptive refinement starting from the coarse mesh and refinement/unrefinement are shown in Figs. 6.26 and 6.27 and Figs. 6.28 and 6.29, respectively. The increase in the maximum error for the Direct Taylor Series Error estimate corresponds to the improvement of the shock resolution. The net effect of the sampling change in this case is that the mean error remains uniform as the refinement progresses for both refinement-only and refinement/unrefinement.

In spite of the fact that the residual is well-defined, the Local Problem Error estimate cannot be used on inviscid problems. Eqn. (4.101) shows that in the case of $\nu = 0$, the local error estimate goes to infinity, rendering the upper bound on the error meaningless.

In the case of supersonic flows, the comparison of the error estimate and the exact error is not relevant – the physical thickness of the shock is infinitely small, compared with the finite thickness in the numerical solution. The role of error estimation is to control the adaptive procedure. Numerical experiments show that any of the proposed error estimates can be used in place of the error indicator Υ , producing equivalent refinement patterns.

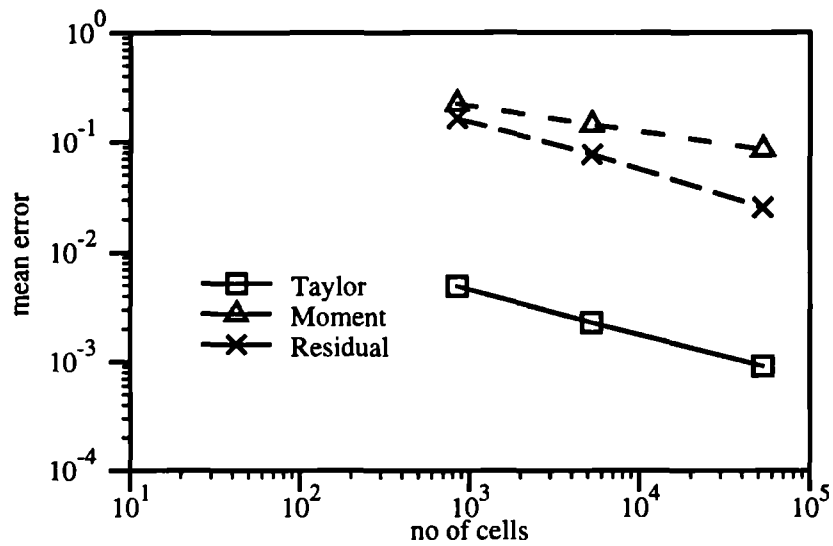


Figure 6.24: Supersonic flow over a forward-facing step: scaling of the mean error for uniform refinement.

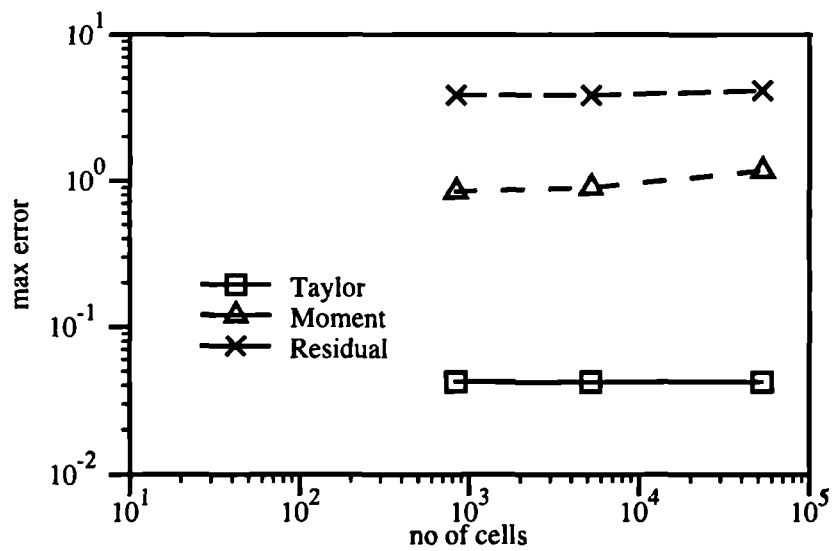


Figure 6.25: Supersonic flow over a forward-facing step: scaling of the maximum error for uniform refinement.

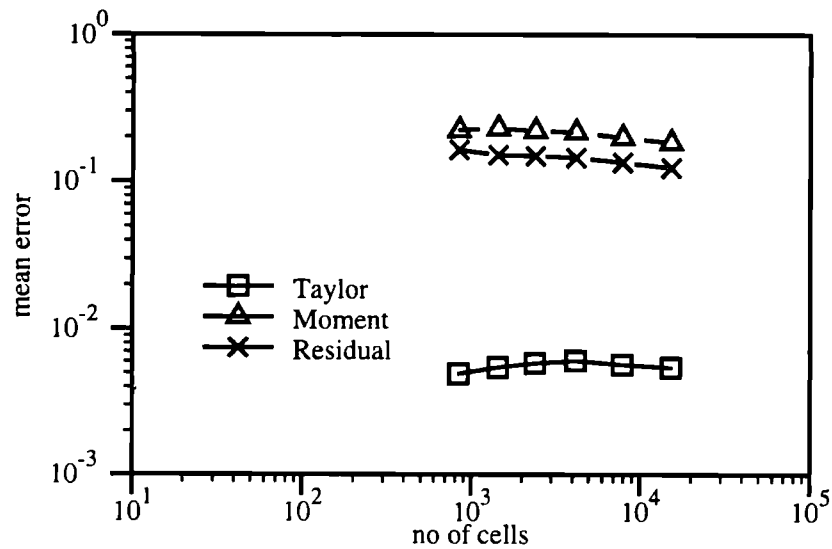


Figure 6.26: Supersonic flow over a forward-facing step: scaling of the mean error for adaptive refinement.

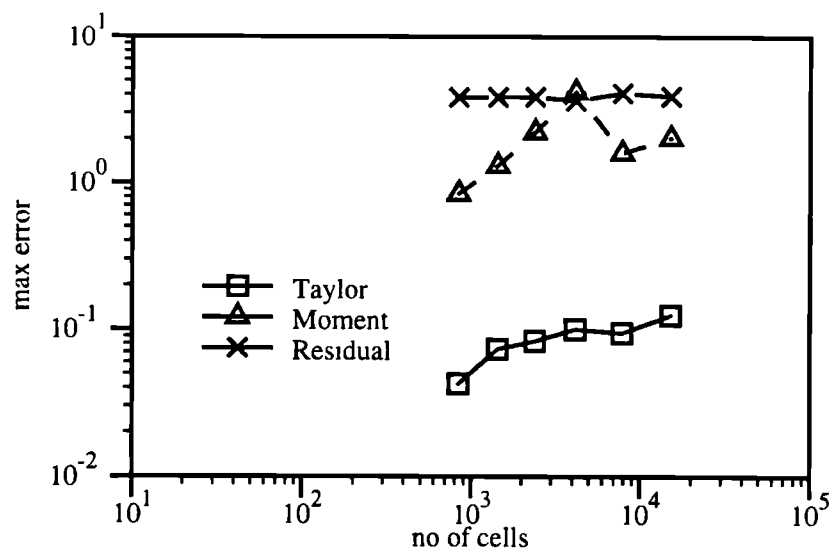


Figure 6.27: Supersonic flow over a forward-facing step: scaling of the maximum error for adaptive refinement.

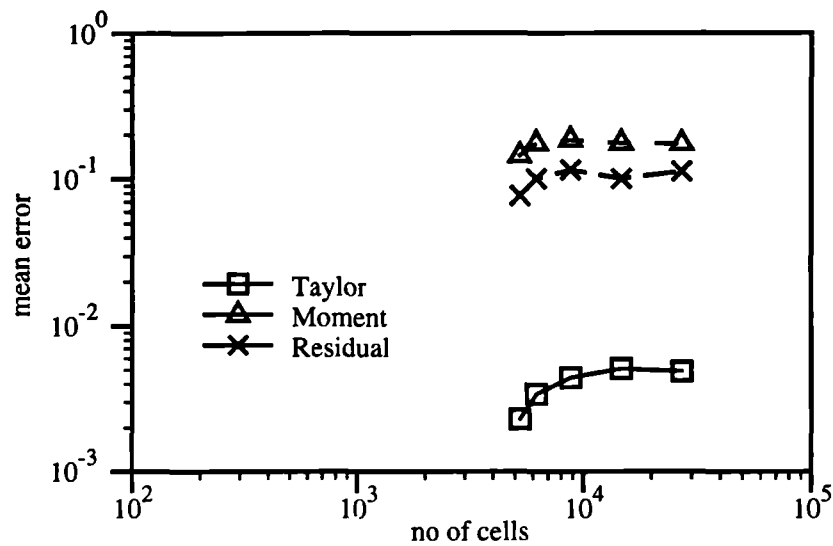


Figure 6.28: Supersonic flow over a forward-facing step: scaling of the mean error for refinement/unrefinement.

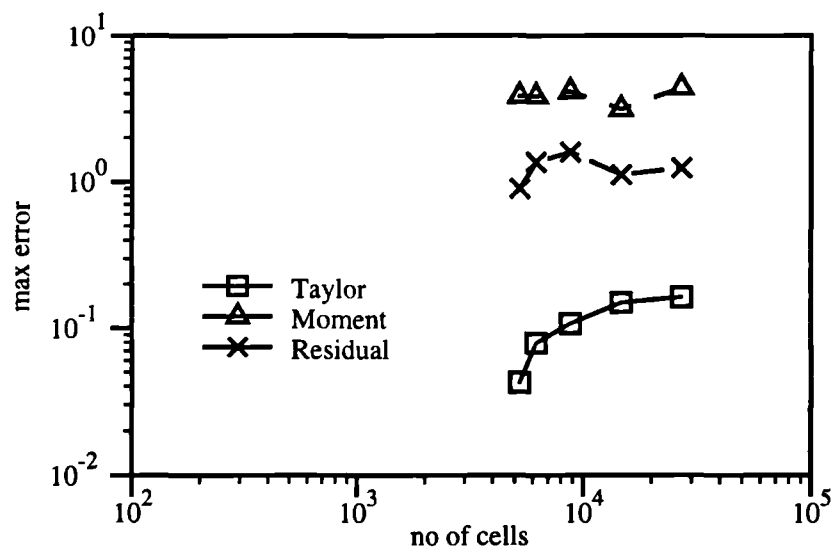


Figure 6.29: Supersonic flow over a forward-facing step: scaling of the maximum error for refinement/unrefinement.

6.3 Laminar and Turbulent Flow Over a 2-D Hill

The geometry of the second test case consists of a symmetric hill in a tunnel, with the blockage ratio of hill height to channel depth of $1/6.07$, Fig. 6.30.

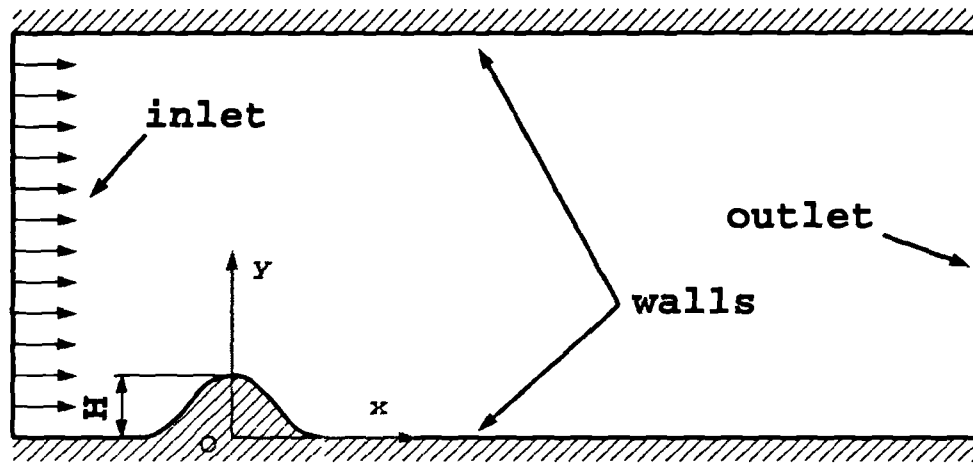


Figure 6.30: Two-dimensional hill test case.

The shape of the hill is defined by the following set of points:

$x [mm]$	0	9	14	20	30	40	54
$y [mm]$	28	27	24	19	11	4	0

Table 6.3: The profile of the hill.

The computational domain starts at $3.6 H$ (or $100 mm$) upstream of the hill summit and extends for $15 H$ downstream. Two flow regimes will be examined:

1. Laminar flow with $Re = 60$, based on the centreline velocity and the hill height. At the inlet boundary, a fully developed parabolic duct flow velocity profile is prescribed.
2. Turbulent flow on $Re = 60000$ Almeida *et al* [5] describe Laser-Doppler anemometer (LDA) measurements of the mean flow and turbulent stresses for these conditions. This test case has also been used in the ERCOFTAC code-comparison exercise [43]. The recommended inlet velocity and turbulence

profiles from the ERCOFTAC workshop are used. The turbulence model selected for this calculation is the non-linear form of the $k - \epsilon$ model by Shih *et al.* [123].

Both flow regimes will be originally solved on four meshes created using transfinite mapping, with 518, 2044, 8584 and 34336 CV-s, respectively. The mesh spacing is slightly graded towards the walls and the hill. A sample mesh with 2044 CV-s is shown in Fig. 6.31.

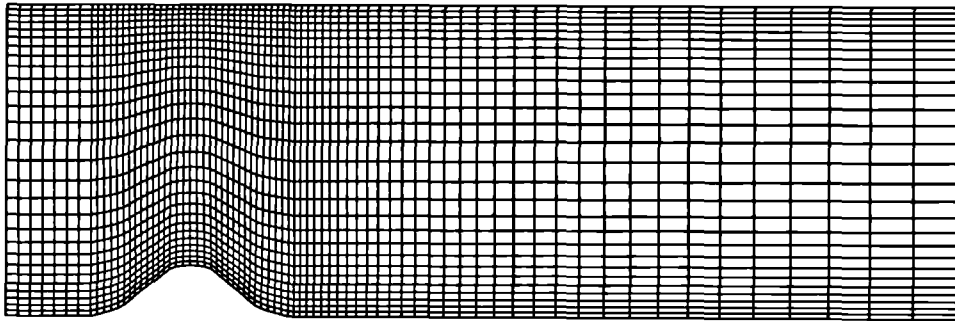


Figure 6.31: Two-dimensional hill: uniform mesh with 2044 CV.

In the second phase, the error-driven adaptive algorithm will be used in two modes: refinement-only from the coarse initial mesh and refinement/unrefinement from the fine mesh.

6.3.1 Laminar Flow

The accuracy of the proposed error estimates can be evaluated only if the exact solution for the problem is available. In the case of two-dimensional laminar flows, it is reasonable to use a numerical solution from a very fine mesh in the place of the analytical solution. The flow over a hill has been resolved on the mesh with 137344 CV-s and the solution has been linearly interpolated to other meshes in order to calculate the “exact” error. Although this number of cells may seem excessive, the “exact numerical” solution is still not considered accurate enough to judge the accuracy of error estimation for the meshes with more than 10000 CV-s.

In the method of solution (see Section 3.8), the pressure equation is used to enforce the continuity constraint on the velocity field. We shall therefore concentrate on the estimation of the velocity error, knowing that appropriate resolution of the velocity field implies accurate pressure distribution.

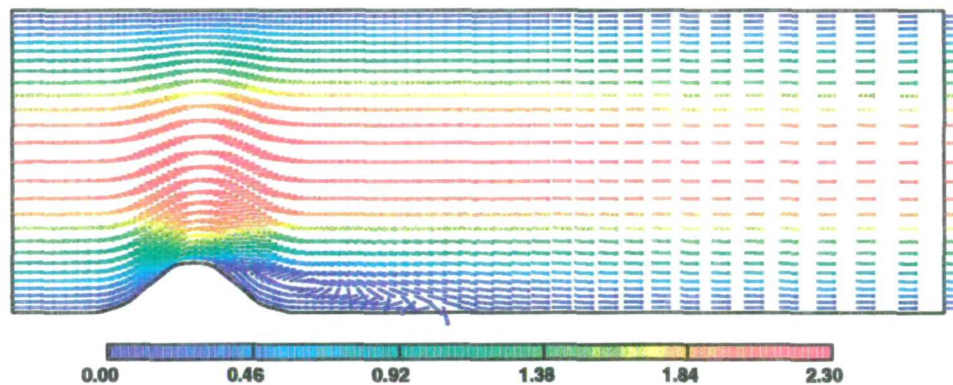


Figure 6.32: Laminar flow over a 2-D hill: velocity field.

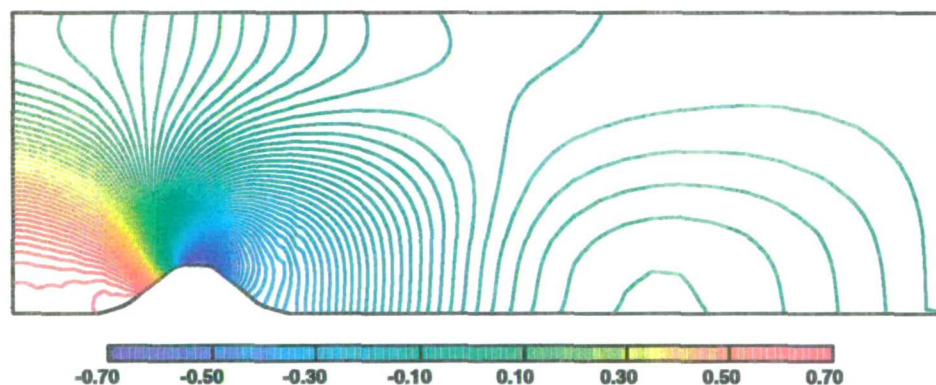


Figure 6.33: Laminar flow over a 2-D hill: pressure field.

The solution obtained on the mesh with 2044 CV-s is shown in terms of velocity and pressure fields in Figs. 6.32 and 6.33.

The difference between the “exact” solution and the velocity field from Fig. 6.32 represents the exact error in the velocity. This error, Fig. 6.34, has a vector form, each component giving the error in the corresponding component of the velocity vector.

An error estimate is set up to highlight the regions of high discretisation error,

rather than to create an error field which will consequently be added to the solution to improve its accuracy. It is therefore more informative to look at the magnitude of the exact error, shown in Fig. 6.35.

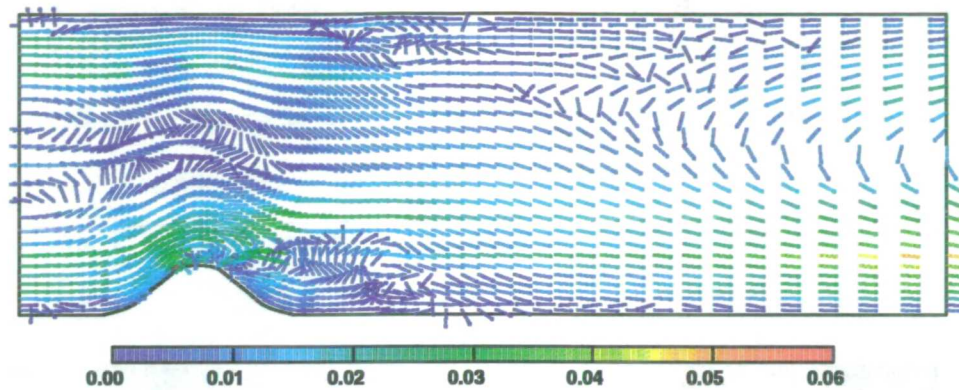


Figure 6.34: Laminar flow over a 2-D hill: vector velocity error, uniform mesh, 2044 CV.

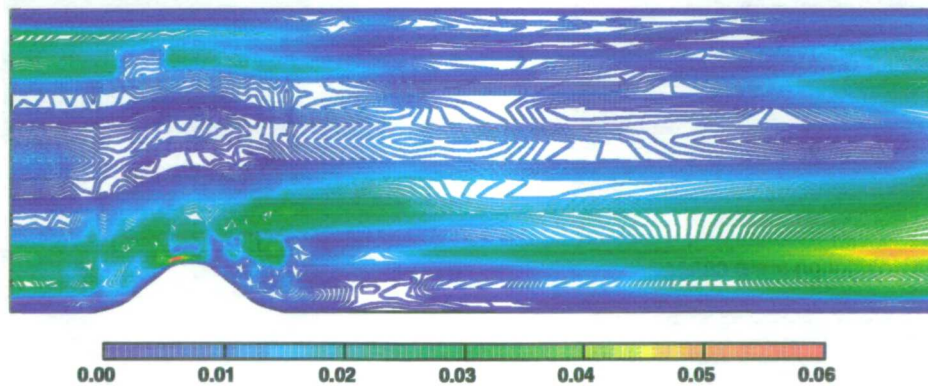


Figure 6.35: Laminar flow over a 2-D hill: velocity error magnitude, uniform mesh, 2044 CV.

Looking at Fig. 6.35, three distinct parts of the exact error can be recognised. The first, upstream of the hill, shows error patterns spreading uniformly from the inlet boundary. The inlet boundary condition corresponds to a fully developed duct flow on the mesh consistent with the number of cells in the y -direction. As the mesh resolution changes, the discrete description of the inlet profile introduces an error in spite of the fact that all inlet conditions represent the same physical situation. The ordered behaviour of this error can also be seen in Fig. 6.34. The error is induced by the description of the inlet boundary condition, which is for the purposes of error

analysis considered to be exact. As a consequence, this error cannot be picked up by an error estimate.

The second region of high error envelops the hill, including the area just above the summit. The error in this region is mainly caused by insufficient mesh resolution close to the summit, with the rest of the error being convected downstream from the peak. The third interesting region lies further downstream, close to the outlet boundary. It is assumed that the error here originates from the accumulated upstream inaccuracies. It is rapidly removed with refinement and disappears on the next finer mesh (see Fig. 6.36).

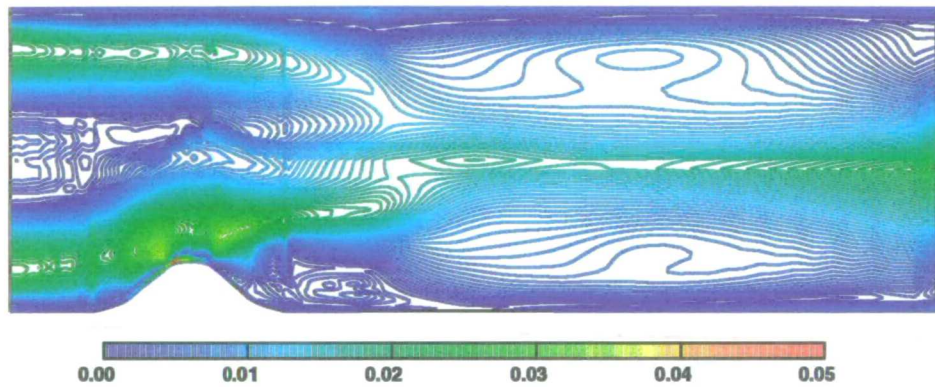


Figure 6.36: Laminar flow over a 2-D hill: velocity error magnitude, uniform mesh, 8584 CV.

The estimated error distributions using the Direct Taylor Series, Moment and Residual Error estimates for the mesh with 2044 CV-s are shown in Figs. 6.37, 6.38 and 6.39. In order to highlight the error distribution rather than its magnitude, the colour-scale in Figs. 6.37, 6.38 and 6.39 has been adjusted.

All three error estimates highlight the front of the hill as the region of high error sources, corresponding to the maximum error in Fig. 6.35. The mesh-induced errors are high at the connection of the orthogonal mesh in front and behind the hill with the region of non-orthogonality above the hill and are visible in all error estimates. Upstream of the hill, the estimated error is low and depends mainly on the mesh spacing. The recirculation zone behind the hill also induces some error, caused by the high velocity gradients between the recirculation bubble and the rest of the flow.

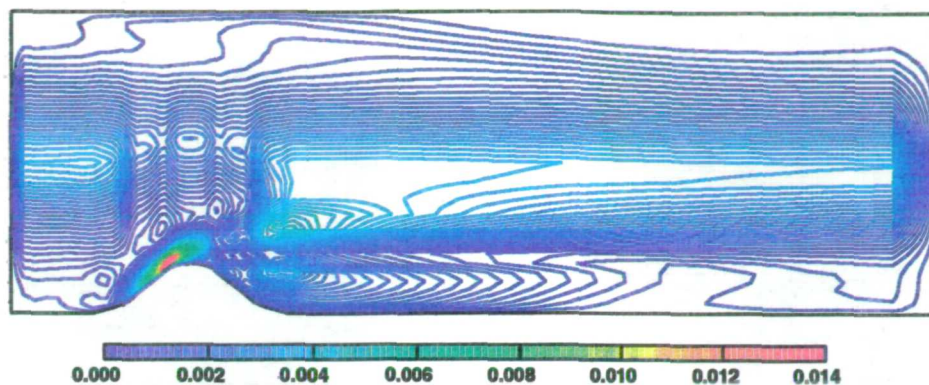


Figure 6.37: Laminar flow over a 2-D hill: Direct Taylor Series Error estimate for the velocity, uniform mesh, 2044 CV.

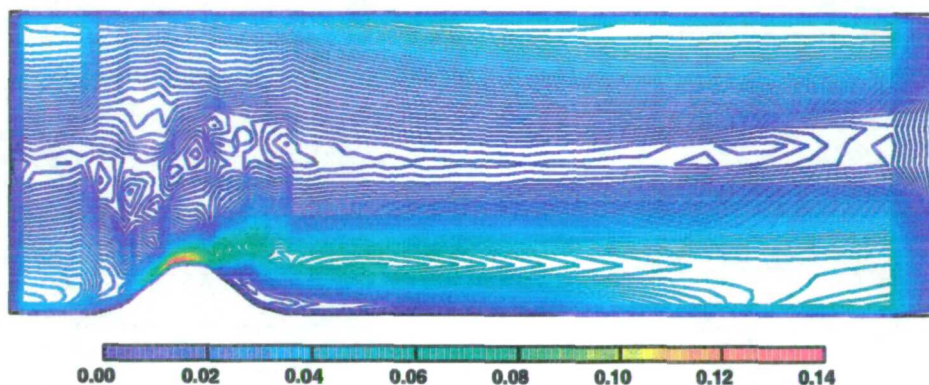


Figure 6.38: Laminar flow over a 2-D hill: Moment Error estimate for the velocity, uniform mesh, 2044 CV.

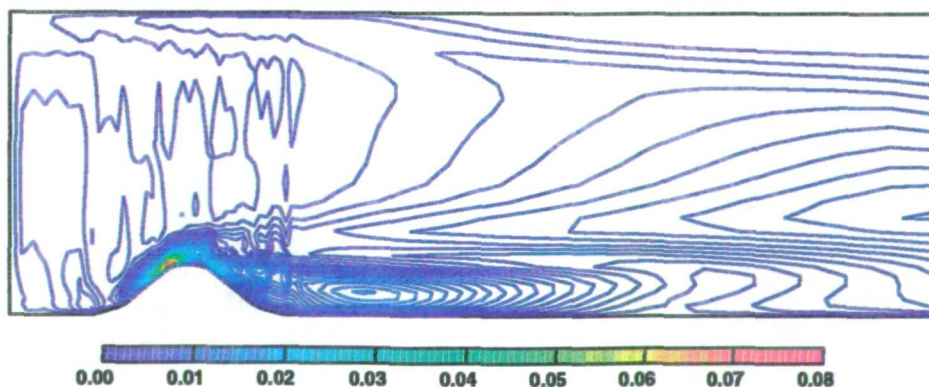


Figure 6.39: Laminar flow over a 2-D hill: Residual Error estimate for the velocity, uniform mesh, 2044 CV.

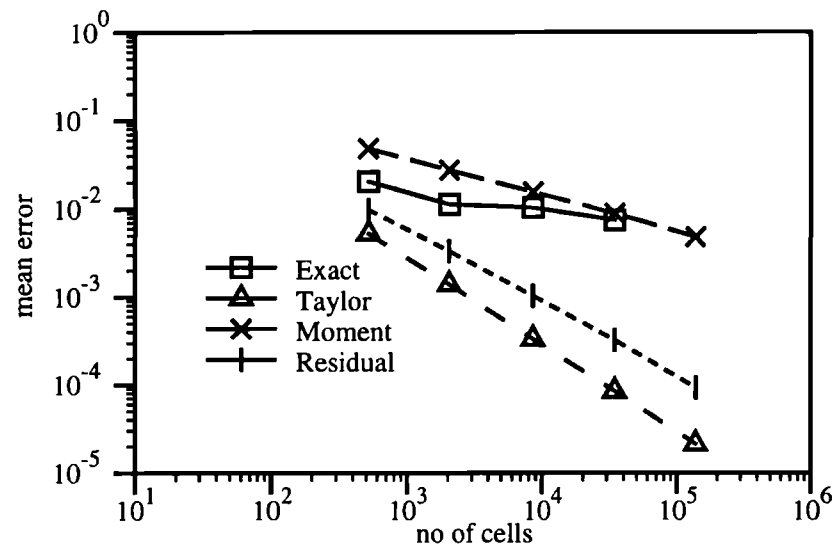


Figure 6.40: Laminar flow over a 2-D hill: scaling of the mean error for uniform refinement.

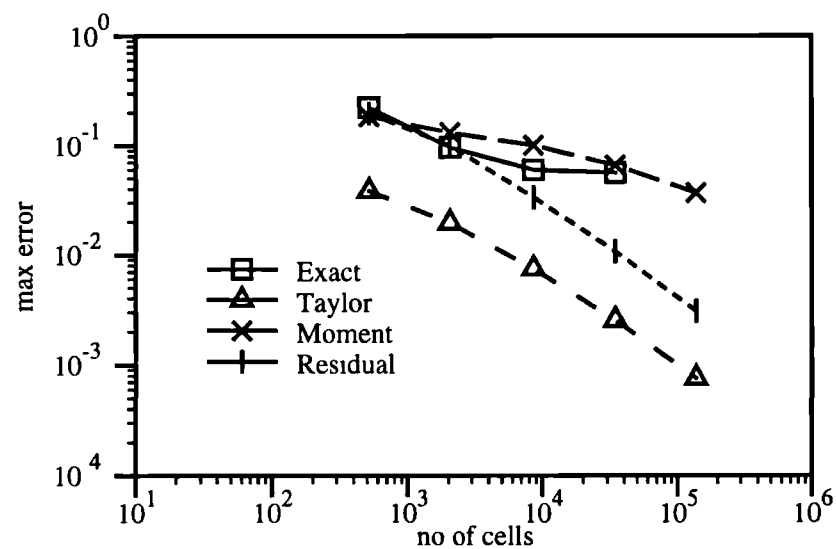


Figure 6.41: Laminar flow over a 2-D hill: scaling of the maximum error for uniform refinement.

The scaling of the mean and maximum for the exact error and the proposed error estimates is given in Figs. 6.40 and 6.41.

On the meshes over 10000 CV-s, the “exact numerical” solution used to obtain the exact error cannot be considered accurate enough in comparison with the solution that is being examined. The maximum error for such meshes is of the order of 0.02 m/s , or 1% of the centreline velocity. The accuracy of the “exact” numerical solution should be at least an order of magnitude better, which with the shown scaling requires a mesh of around 1000000 CV-s. The relative inaccuracy of the “exact numerical” solution causes a distortion in the scaling of the exact error with mesh refinement.

The Moment and Residual Error estimates capture the mean and maximum error with reasonable accuracy, particularly on coarse meshes. The Direct Taylor Series Error estimate underpredicts its magnitude, which is consistent with expectations. The influence of the neglected fourth-order terms can be seen in the scaling of the mean Moment Error. Having in mind that the average error on the mesh with 8584 CV-s is around 0.5 % of the centreline velocity, it seems plausible that the neglected fourth-order terms influence the accuracy of the error estimate.

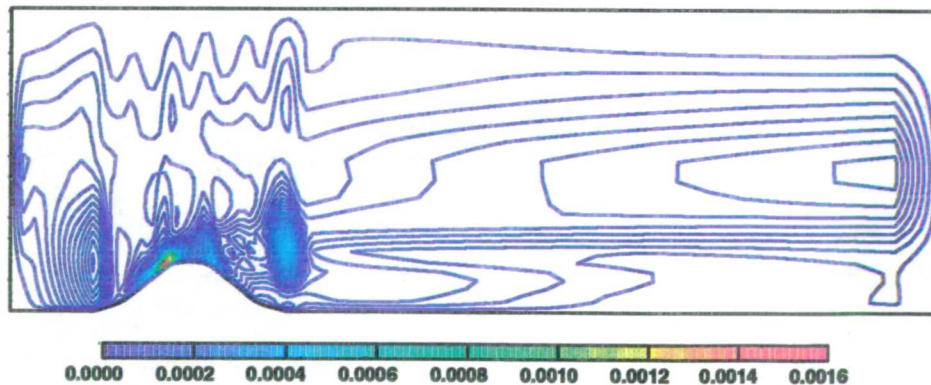


Figure 6.42: Laminar flow over a 2-D hill: estimated error norm, uniform mesh, 2044 CV.

The distribution of the estimated error norm according to the Local Problem Error estimate is shown in Fig. 6.42. This correctly highlights the region of high velocity gradients close to the summit of the hill. As a consequence of the high

local residual, this error norm also pinpoints the mesh-induced errors behind the hill. Following Oden *et al.* [101, 102], the error norm from the Local Problem Error estimate can be used in an alternative way. The formulation of the error norm corresponds to the magnitude of the error in the quantity that is being minimised by the numerical solution procedure. In the case of the laminar flow, this is the total dissipation in the system, defined as:

$$\Phi_{tot} = \int_V \Phi dV = \int_V \nu [\nabla U + (\nabla U)^T] : \nabla U dV \quad (6.2)$$

The accuracy of the solution can therefore be measured in terms of the ratio of $(e, E)_*^2$ and Φ_{tot} , Fig. 6.43, corresponding to the relative error in the dissipation. As a guideline, the “exact” error in Φ_{tot} can be calculated using the dissipation from the “exact numerical” solution. The Local Problem Error estimate guarantees that

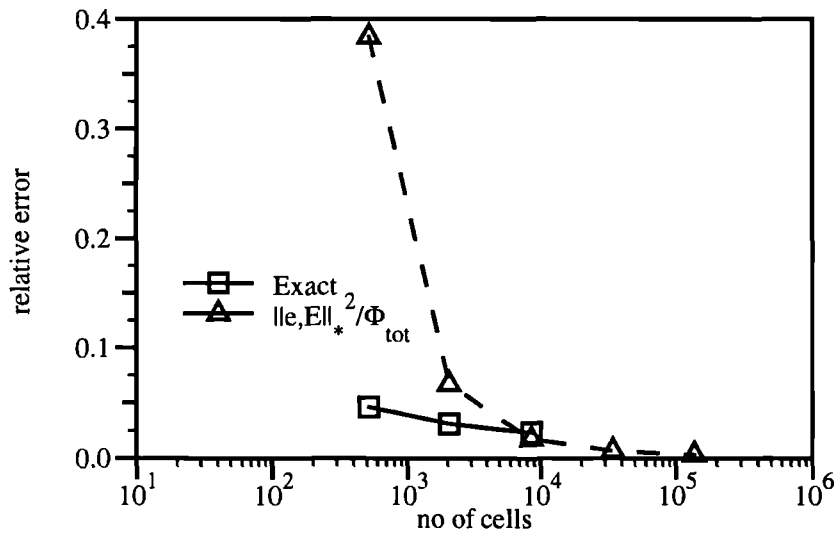


Figure 6.43: Laminar flow over a 2-D hill: scaling of the relative error in dissipation for uniform refinement.

$(e, E)_*^2$ provides the upper bound on the dissipation error, which is not the case for the final point in Fig. 6.43. This can be attributed partly to the inaccuracy of the “exact” numerical solution and partly to the quality of the solution procedure for the local error problems.

In the second phase, two refinement strategies will be used. refinement-only from the initial mesh with 2044 CV-s, Fig. 6.31 and refinement unrefinement from the

mesh with 8584 CV-s. The Residual Error estimate is used to highlight the sources of the error, with the refinement/unrefinement parameters set to:

$$\lambda_{ref} = 1.5$$

$$\lambda_{unref} = 0.5$$

$$\xi = 0.6$$

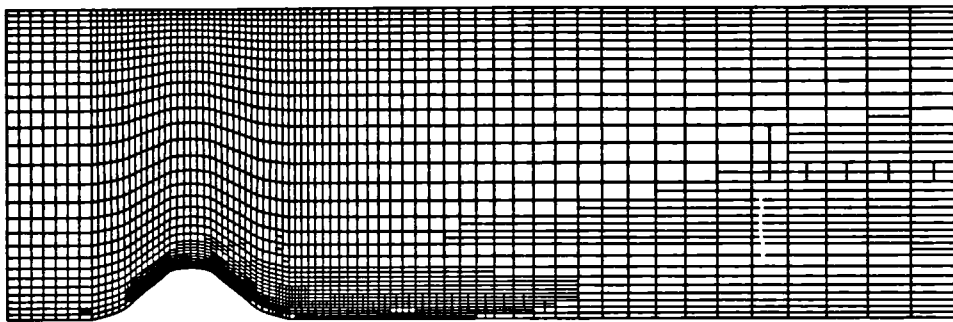


Figure 6.44: Laminar flow over a 2-D hill: mesh after the first level of refinement.

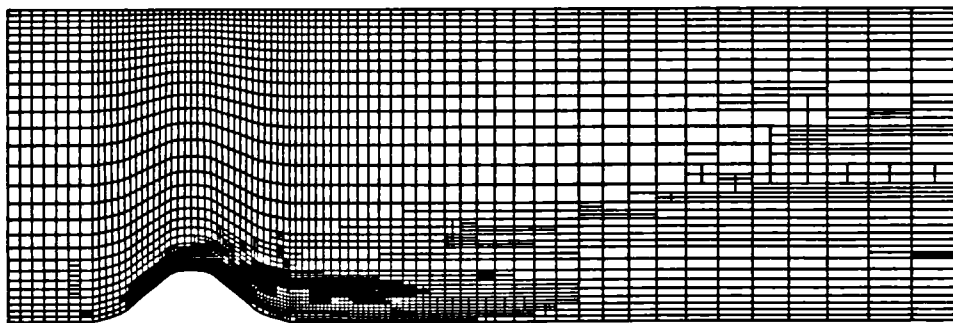


Figure 6.45: Laminar flow over a 2-D hill: mesh after the second level of refinement.

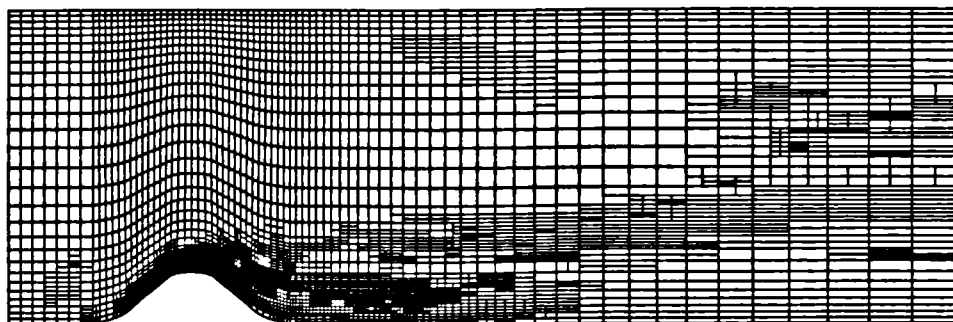


Figure 6.46: Laminar flow over a 2-D hill: mesh after the third level of refinement.

The mesh changes in the first three levels of refinement starting from the coarse mesh are shown in Figs. 6.44, 6.45 and 6.46. The cells are concentrated around the summit and in the recirculation region behind the hill. Some cells are also added close to the outlet boundary in the middle of the channel.

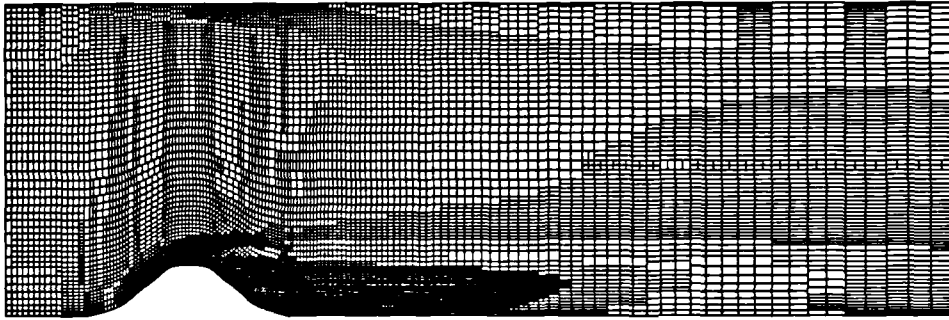


Figure 6.47: Laminar flow over a 2-D hill: first level of refinement/unrefinement.

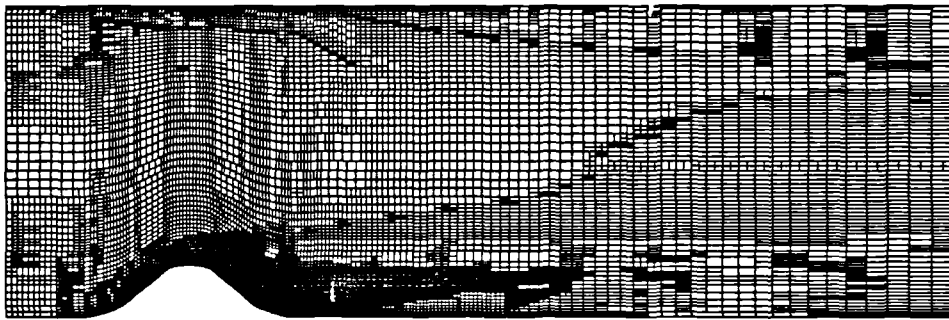


Figure 6.48: Laminar flow over a 2-D hill: second level of refinement/unrefinement.

The refinement/unrefinement algorithm behaves in a similar way: the mesh changes in the early stages of refinement/unrefinement are shown in Figs. 6.47 and 6.48. Refinement is again concentrated around the hill in order to resolve the region of high error at the front of the hill and in the recirculation zone. Another zone of refinement extends downstream from the summit towards the middle of the channel. This is the result of the interaction between the velocity field (Fig. 6.32) and the grading in the initial mesh. The mesh spacing is initially graded towards the wall. If one assumes that the influence of the hill does not extend all the way to the outlet boundary, the velocity field close to the outlet corresponds to the fully developed duct flow, with the parabolic velocity profile. The second derivative of U in the

y -direction is constant, as a result of which the refinement procedure tends to create a more uniform mesh by refining the cells in the centre of the channel.

The interaction between the mesh changes and the estimated error distribution shown in Figs. 6.49, 6.50 and 6.51 gives a useful insight into the development of the adaptive solution. The first level of refinement creates a rapid drop in the maximum error (distribution of the estimated error on the initial mesh is given in Fig. 6.39; note the change in the colour-scale). The peak error is still located close to the summit of the hill. The second level of refinement moves the maximum error away from the hill, into the area of high mesh grading and non-orthogonality. Several new regions of high error are detected in the second and third level of refinement, created as a consequence of relatively low local mesh quality. In spite of the fact that the initial error caused by insufficient mesh resolution is removed very rapidly, further improvement in accuracy is limited by the mesh grading and non-orthogonality. The quality of the mesh is an important issue in the case of adaptive laminar calculations.

The situation is even more clear in the case of refinement/unrefinement starting from the mesh with 8584 CV-s. Fig. 6.52 presents the distribution of the Residual Error estimate on the initial (uniform) mesh. The changes in the estimated error in first two levels of refinement are shown in Figs. 6.53 and 6.54, corresponding to the meshes from Figs. 6.47 and 6.48. Additional mesh resolution around the summit completely removes the local peak in the error. The cells removed through unrefinement close to the upper boundary in the first level cause a high error (see Fig. 6.53) and are partially re-introduced when the adaptive procedure is repeated, Fig. 6.48. Generally, the main error sources in the adaptively refined meshes are created in the area of interaction of the coarse and fine mesh. In this case, unrefinement causes more problems in terms of mesh quality than is reasonably acceptable, considering the limited reduction in the number of cells.

Laminar flows regularly create smooth solutions in which only a part of the error can be attributed to the insufficient resolution. The mesh quality plays an important role in the overall solution accuracy. The best performance from the adopted mesh refinement algorithm is obtained when only two or three levels of adaptive re-

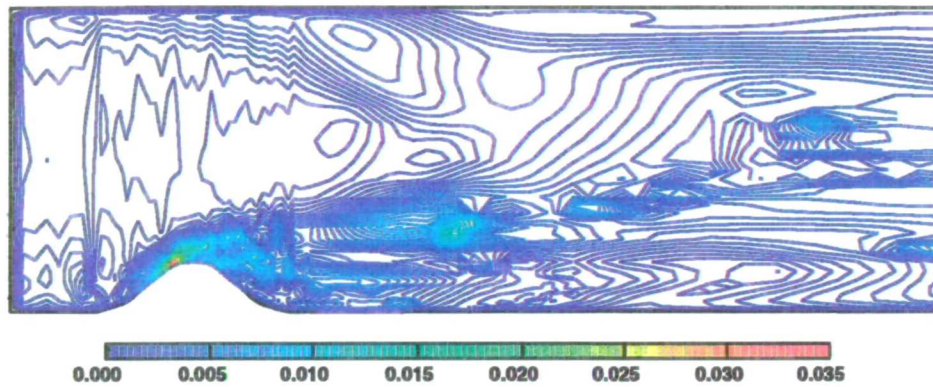


Figure 6.49: Laminar flow over a 2-D hill: Residual Error estimate after the first level of refinement.

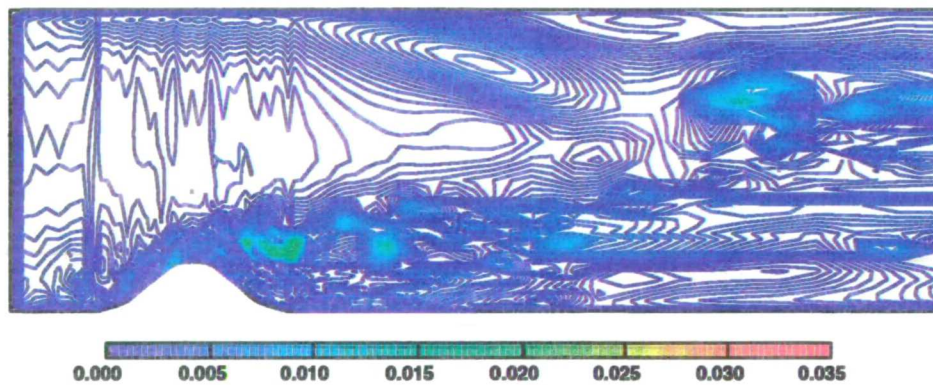


Figure 6.50: Laminar flow over a 2-D hill: Residual Error estimate after the second level of refinement.

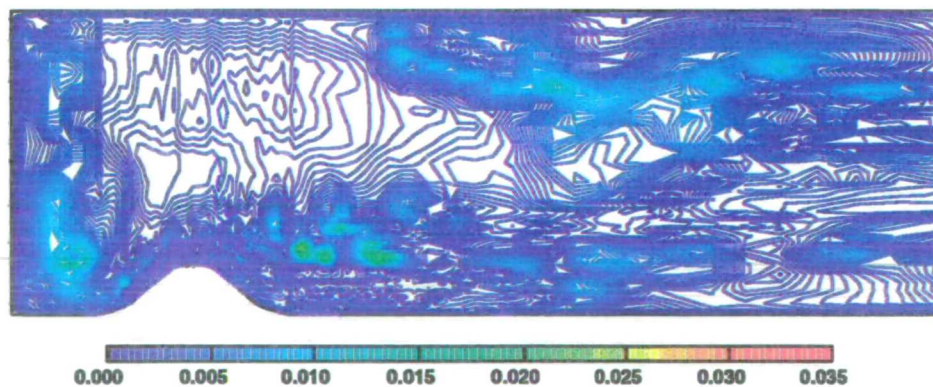


Figure 6.51: Laminar flow over a 2-D hill: Residual Error estimate after the third level of refinement.

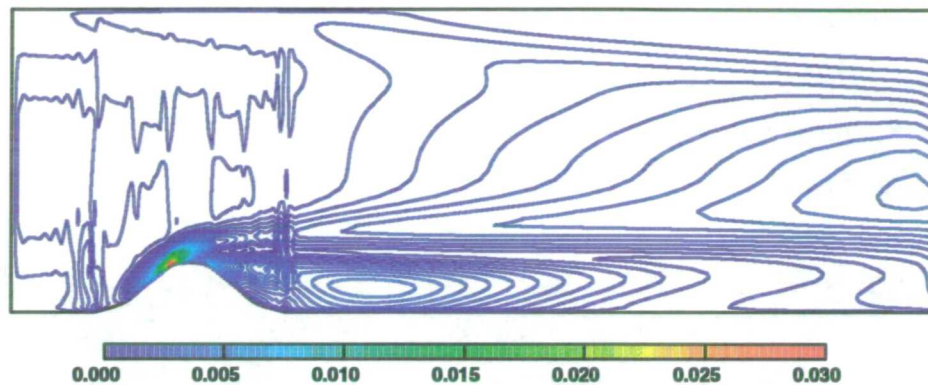


Figure 6.52: Laminar flow over a 2-D hill: Residual Error estimate on the uniform mesh, 8584 CV-s.

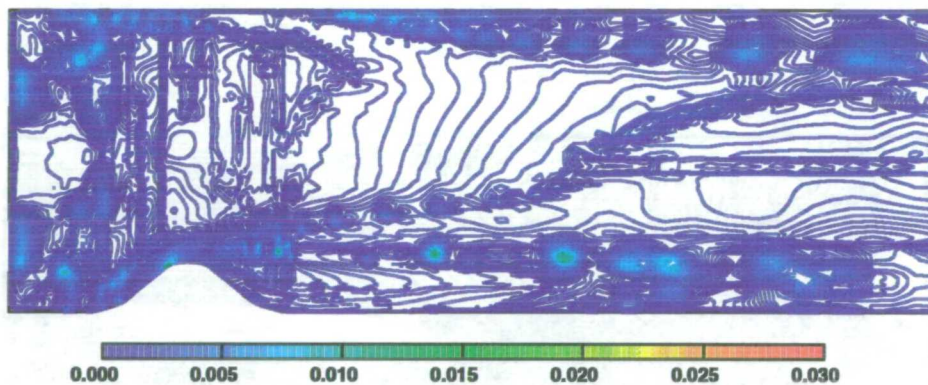


Figure 6.53: Laminar flow over a 2-D hill: Residual Error estimate after the first level of refinement/unrefinement from the fine mesh.

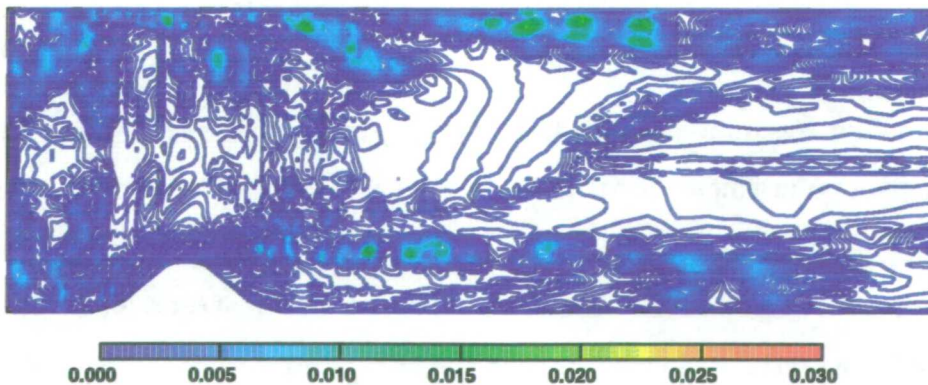


Figure 6.54: Laminar flow over a 2-D hill: Residual Error estimate after the second level of refinement/unrefinement from the fine mesh.

finement are used. The unrefinement procedure introduces mesh distortions of such significance that its use cannot be justified in this case. In order to demonstrate this fact, a refinement-only calculation starting from the mesh with 8584 CV-s has been performed. The mesh after two levels of refinement is shown in Fig. 6.55, with the

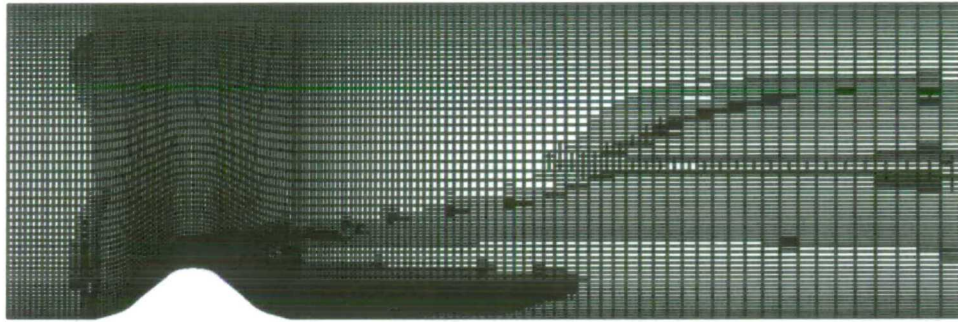


Figure 6.55: Laminar flow over a 2-D hill: second level of refinement-only from the fine mesh.

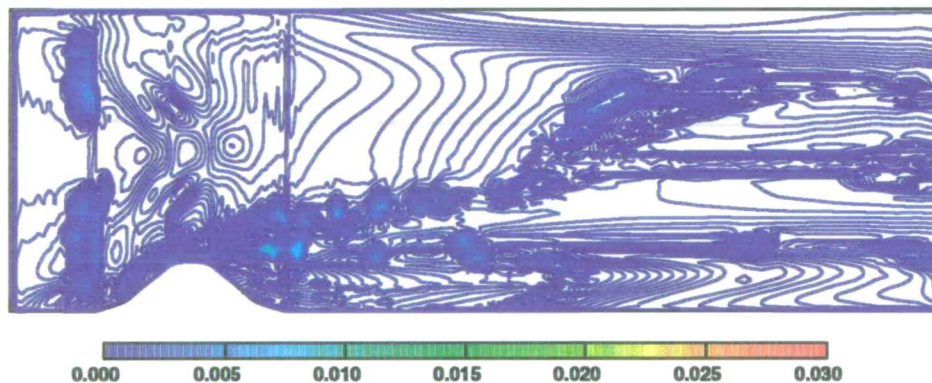


Figure 6.56: Laminar flow over a 2-D hill: Residual Error estimate after the second level of refinement/unrefinement.

corresponding Residual Error distribution in Fig. 6.56. The corresponding results for refinement/unrefinement are given in Figs. 6.48 and 6.54. Refinement-only does not degrade the quality of the mesh close to the top wall and the estimated error levels are considerably lower than in the case of refinement/unrefinement.

Finally, the estimated error levels in the adaptively refined meshes will be examined. Unfortunately, the “exact numerical” solution cannot be used to measure the accuracy of the error estimates. Mesh adaptivity creates refinement patches that

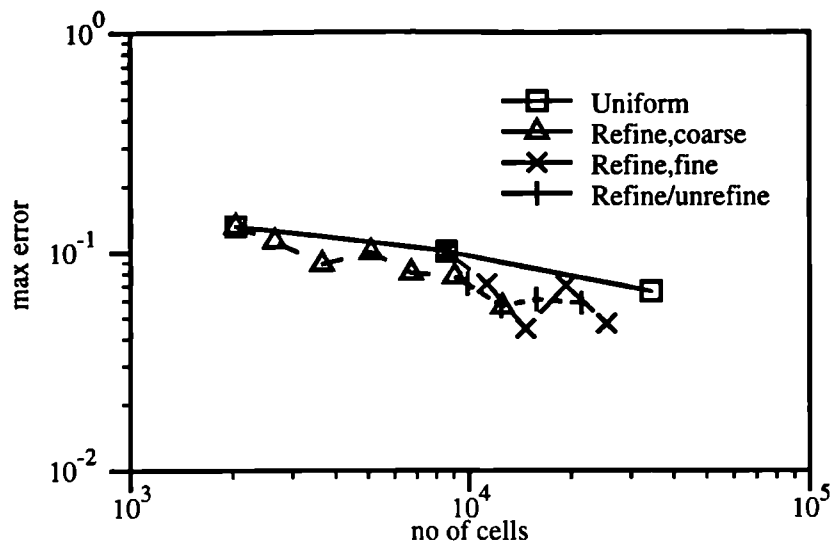


Figure 6.57: Laminar flow over a 2-D hill: scaling of the maximum Moment Error for different types of refinement.

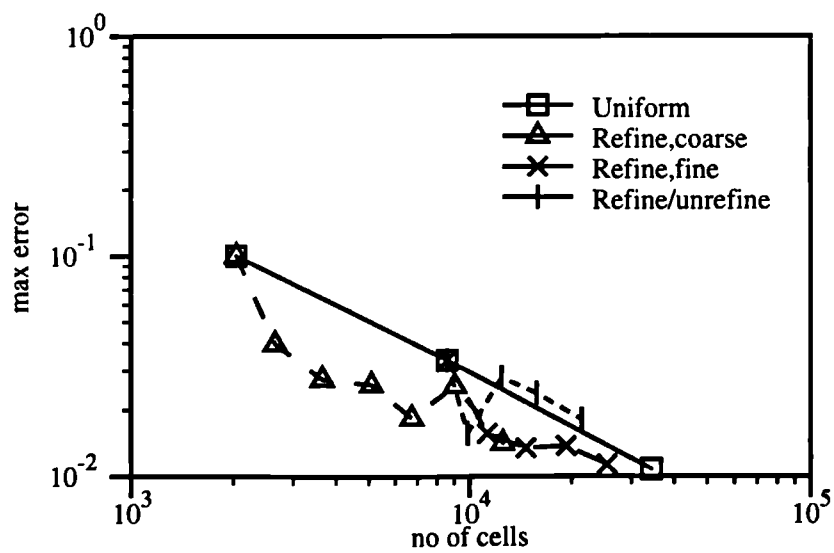


Figure 6.58: Laminar flow over a 2-D hill: scaling of the maximum Residual Error for different types of refinement.

very quickly become as fine or finer than the mesh used for the “exact numerical” solution, rendering it unusable for our purposes. It is therefore more interesting to examine the performance of different adaptive procedures in terms of the reduction of the estimated error.

Figs. 6.57 and 6.58 show the change in the error with the number of cells on adaptively refined meshes. The adaptive procedure reduces the error faster than uniform refinement according to both the Moment and Residual Error estimates, although the gain seems to be much smaller than in the supersonic test case from Section 6.2. Between the two estimates, the drop is faster for the Residual Error, as it is used to mark the cells for refinement.

The refinement-only procedure starting from the coarse initial mesh consistently performs better than uniform refinement in spite of the reduced mesh quality. If we limit the refinement to only three levels, all refinement strategies perform reasonably well.

The error reduction shown in Fig. 6.58 does not considerably change if the Direct Taylor Series or Moment Error estimates are used to mark the cells for refinement. The largest change in the performance can be made by changing the refinement parameters (λ_{ref} , λ_{unref} and ξ), as they influence the quality of the refined mesh. Generally, lower ξ creates smoother grading and improves the mesh quality by preserving the original aspect ratio of the cell. At the same time, it also increases the number of cells in the mesh through the “1-irregularity” principle, thus increasing the cost of the computation. When specifying the refinement and unrefinement thresholds, one should have in mind the desired accuracy relative to the accuracy of the initial solution, as well as the importance of the mesh quality. It has been shown that the cells initially removed through unrefinement were re-introduced in the next level of refinement. This is typical for a large number of refinement levels on smoothly changing solutions and should be avoided, as it causes a potential reduction of the mesh quality.

The performance of the adaptive procedure in laminar flows is critically dependent on the quality of the refined mesh. An adaptive mesh generation able to create

smoothly graded meshes with low non-orthogonality is expected to perform considerably better than the proposed local mesh refinement algorithm in terms of error reduction. As an automatic mesh generator with such properties is not available, the only alternative is to limit the refinement to a modest number of embedded levels. This conclusion is, however, somewhat influenced by the typical error levels in the solution, which are of the order of 0.5 % or less. If the flow structures were more complex, the adaptive procedure would perform considerably better. On the other hand, the mesh resolution needed to produce the “exact numerical” solution would also be considerably higher and it would be impossible to judge the accuracy of the error estimates.

6.3.2 Turbulent Flow

The complexity of turbulent flows presents an even more challenging problem for both error estimation and adaptive refinement. In this test case, the non-linear $k - \epsilon$ turbulence model by Shih *et al.* [123] is used to simulate the turbulent flow over a hill. In order to preserve boundedness in the k and ϵ transport equations, the Gamma differencing scheme is used. Since the velocity is not a bounded property, the convection term in the momentum equation is discretised using Central Differencing.

The fields of velocity, turbulent kinetic energy and its dissipation for the mesh with 2044 CV-s are shown in Figs. 6.59, 6.60 and 6.61. The main feature of the flow is the recirculation bubble behind the hill, with high gradients in all fields. As in the case of laminar flow, high velocity gradients also occur near the summit, due to the acceleration of the flow and the compression of the boundary layer. For this test case, the measured inlet velocity profile does not correspond to the fully developed turbulent duct flow. The developing boundary layer along the top wall can be seen in the k field, Fig. 6.60. It is interesting to note the local peak in ϵ at the detachment point on the lee slope of the hill and its rapid variation close to the wall around the reattachment point.

Figs. 6.59, 6.60 and 6.61 illustrate that every variable of the coupled system

requires high resolution in a different part of the domain.

At this point, it is necessary to examine the interaction between error estimation and the treatment of the wall used in this calculation. The near-wall region of high gradients is bridged by the wall-functions, which creates an additional term in the momentum equation in order to compensate for the increased shear stress at the wall. The additional drag is treated as a change in the effective viscosity at the “wall face”, carrying the difference between the assumed linear and the logarithmic velocity profile between the cell centre and the wall. For the k and ϵ transport equations, the situation is somewhat different: wall-functions use the local equilibrium assumption and prescribe the generation of k and the value of ϵ in the near-wall cell.

Each of the error estimates reacts differently to the wall treatment. In the momentum equation, the Direct Taylor Series and Residual Error estimates recognise the high velocity gradients in near-wall cells and report high local errors. Both of these error estimates use the local velocity gradient and cannot fully recognise the wall-function modification. Although it is not clear how to judge the accuracy of the velocity in the near-wall cell, having in mind that it compensates for the proximity of the wall, the estimated error can still be useful. The error estimates take into account the bulk of the flow when estimating the error and therefore offer the information on the “relative inaccuracy” of the wall-function treatment along the wall.

Error estimation for the k transport equation does not require any modification: wall-functions introduce a near-wall generation term which does not require any special treatment.

The estimation of the error in ϵ needs to compensate for the fact that the wall-function treatment prescribes the value of ϵ in the near-wall cell. This value is not consistent with the solution of the ϵ equation in the bulk of the domain and the error in the near-wall cell is therefore considered to be zero. The near-wall ϵ interacts with the rest of the domain through the transport terms of the ϵ equation, causing high gradients near to the wall. In spite of the near-wall treatment, the original formulation of the error estimates is still valid in the bulk of the flow

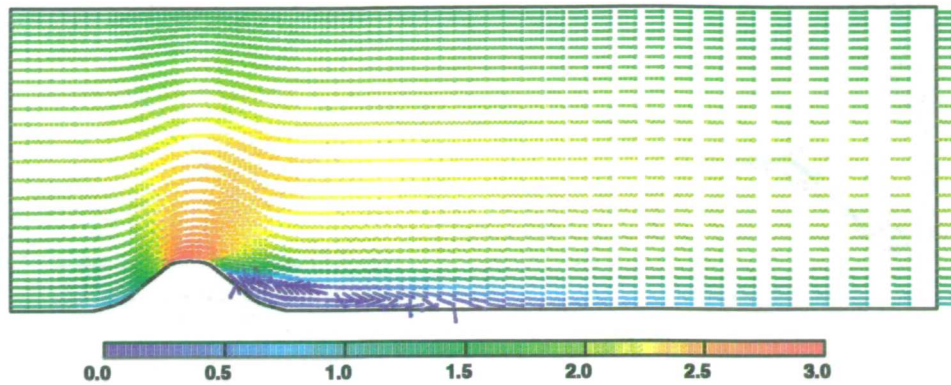


Figure 6.59: Turbulent flow over a 2-D hill: velocity field, uniform mesh, 2044 CV.

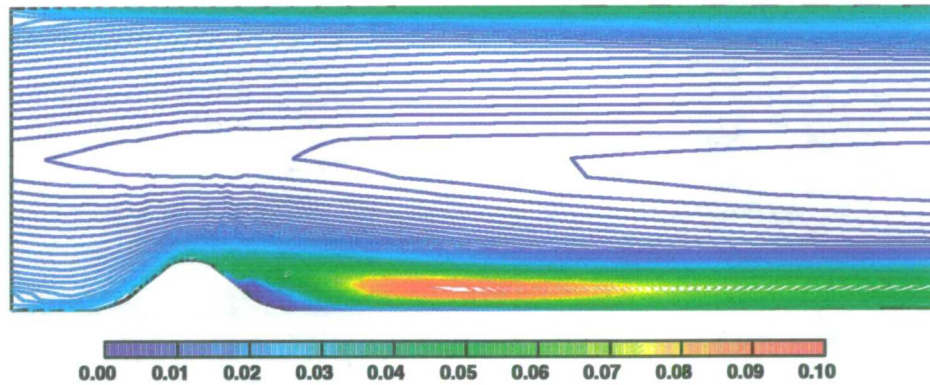


Figure 6.60: Turbulent flow over a 2-D hill: turbulent kinetic energy field, uniform mesh, 2044 CV.

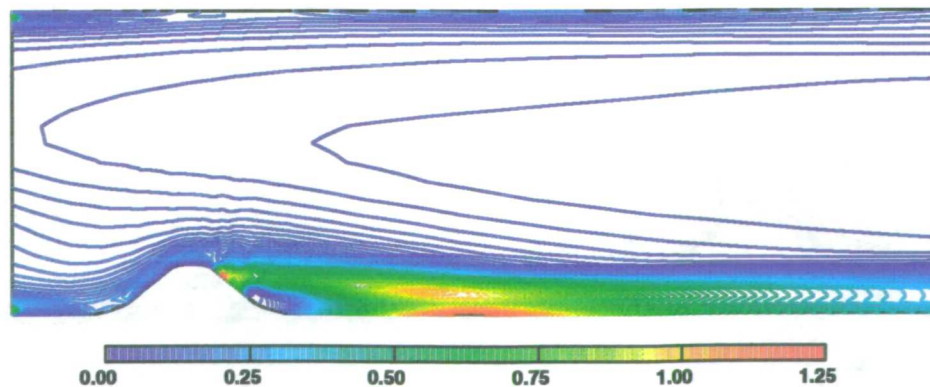


Figure 6.61: Turbulent flow over a 2-D hill: dissipation of turbulent kinetic energy, uniform mesh, 2044 CV.

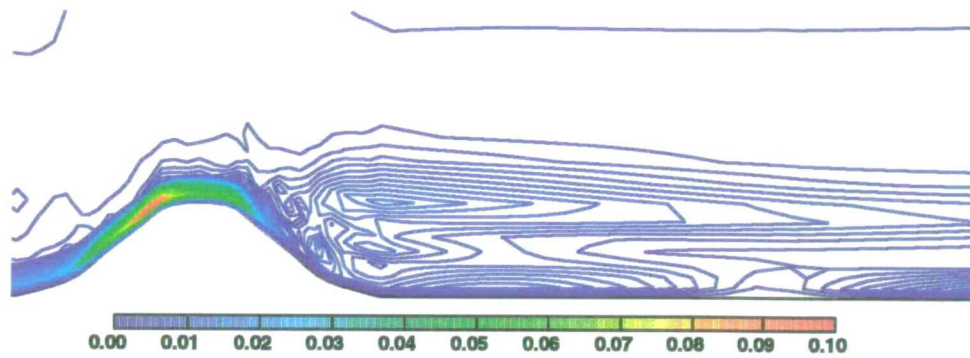


Figure 6.62: Turbulent flow over a 2-D hill: Direct Taylor Series Error estimate for velocity, uniform mesh, 2044 CV.

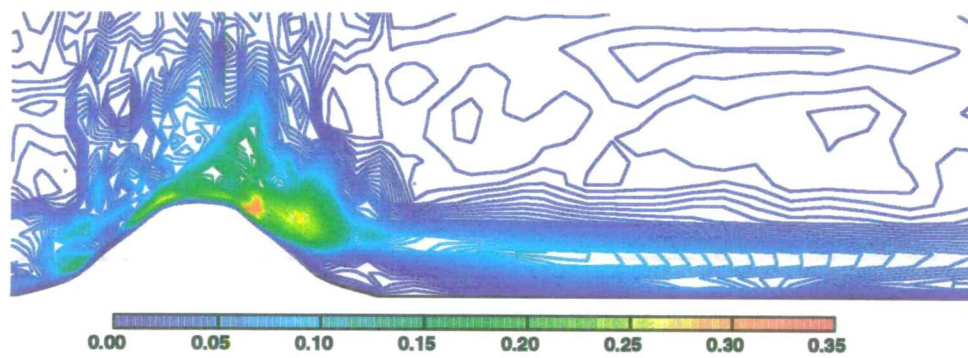


Figure 6.63: Turbulent flow over a 2-D hill: Moment Error estimate for velocity, uniform mesh, 2044 CV.

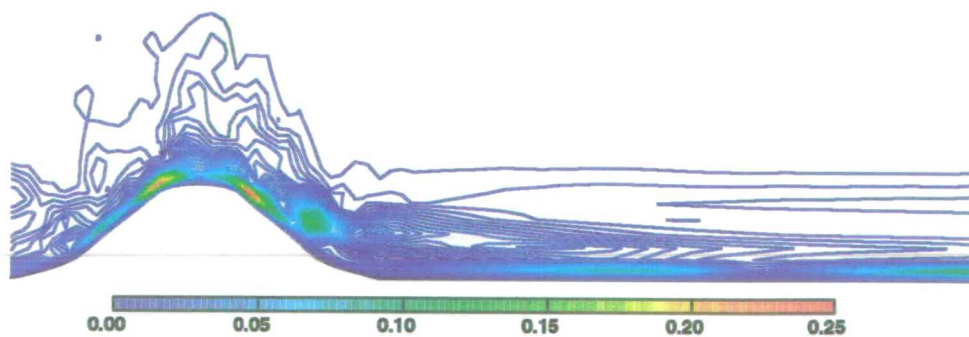


Figure 6.64: Turbulent flow over a 2-D hill: Residual Error estimate for velocity, uniform mesh, 2044 CV.

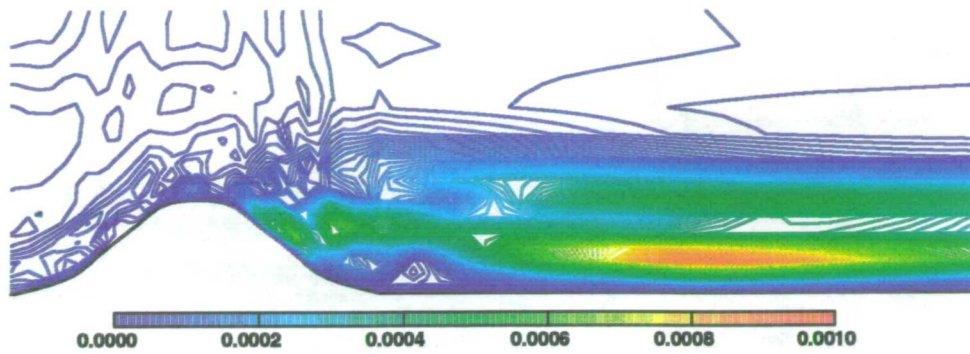


Figure 6.65: Turbulent flow over a 2-D hill: Direct Taylor Series Error estimate for k , uniform mesh, 2044 CV.

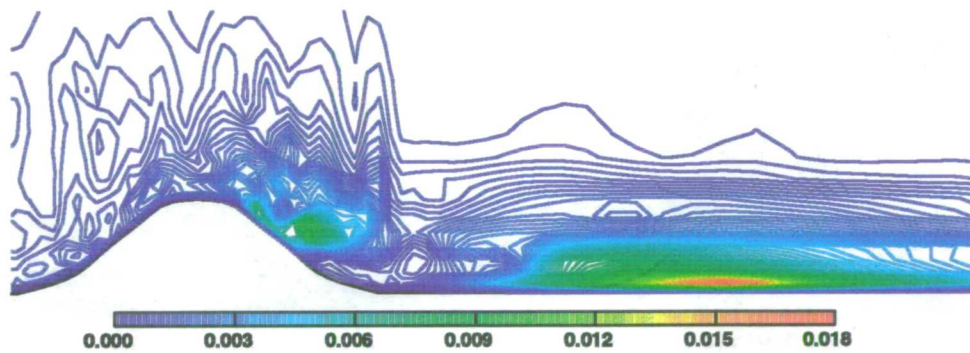


Figure 6.66: Turbulent flow over a 2-D hill: Moment Error estimate for k , uniform mesh, 2044 CV.

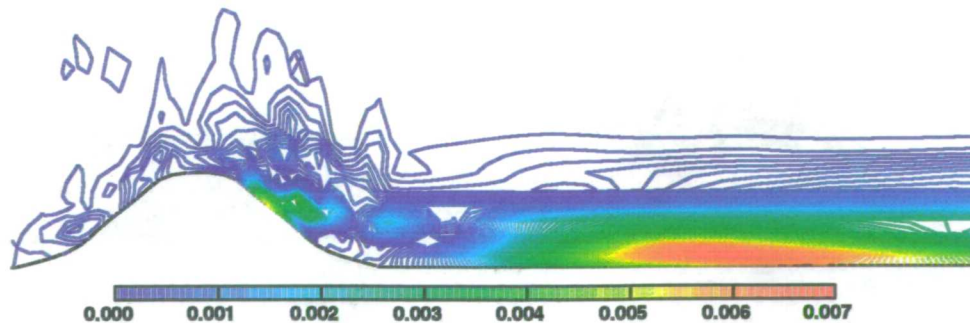


Figure 6.67: Turbulent flow over a 2-D hill: Residual Error estimate for k , uniform mesh, 2044 CV.

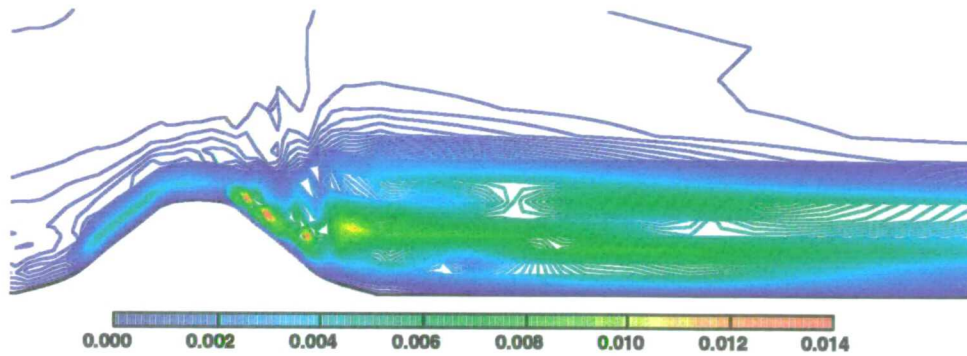


Figure 6.68: Turbulent flow over a 2-D hill: Direct Taylor Series Error estimate for ϵ , uniform mesh, 2044 CV.

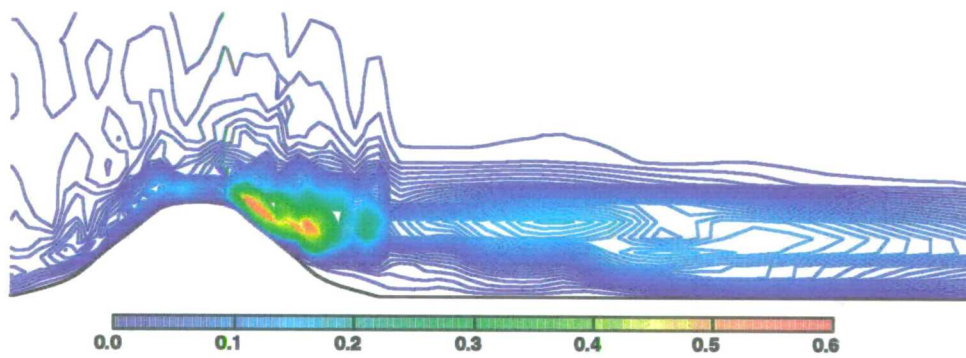


Figure 6.69: flow over a 2-D hill: Moment Error estimate for ϵ , uniform mesh, 2044 CV.

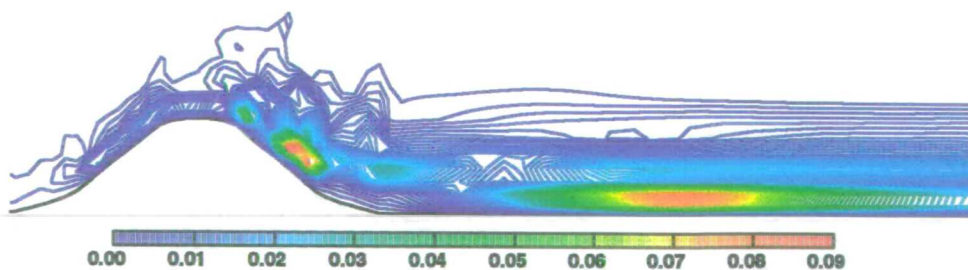


Figure 6.70: Turbulent flow over a 2-D hill: Residual Error estimate for ϵ , uniform mesh, 2044 CV.

It is possible to establish the level of the modelling error for the wall-functions, based on the value of y^+ . In spite of the fact that wall-functions are in principle able to cover the whole range of y^+ , from the laminar sublayer to the log-law layer, it is well-known that they produce the best results for $50 < y^+ < 100$. The estimate of the near-wall accuracy and consequently the refinement criterion can therefore be based on y^+ condition.

Figs. 6.62, 6.63 and 6.64 present the distribution of the Direct Taylor Series, Moment and Residual Error estimates for the velocity field. The error estimates highlight several regions of high error: the windward slope of the hill, the point of flow detachment on the lee slope and the recirculation bubble. The Direct Taylor Series Error estimate does not pick up the high error around detachment. Here, the complex flow field creates a rapid variation in both the magnitude and the direction of the velocity. The actual velocity magnitude is relatively small and the mesh is too coarse to detect the local peak in the second gradient of the velocity used to estimate the error. As the mesh becomes finer, the second gradient calculated on the discrete mesh becomes closer to its exact value, resulting in a more accurate error estimation. This is visible in the scaling of the maximum error with the number of cells for uniform refinement, which will be shown later. On the windward slope, the compression of the boundary layer increases the second velocity gradient, resulting in the high local error. Here, the flow is aligned with the mesh and does not change direction, making it easier to resolve the second velocity gradient. The Moment and Residual Error estimate recognise both the compression of the boundary layer and the detachment point as the regions of the high error.

The variation in the accuracy of the solution near the wall can be seen in Fig. 6.64. The error in the near-wall cell downstream from the hill shows the dependence on the velocity magnitude. If the local velocity is low (just behind the hill or at the reattachment point), the error is smaller. The Moment Error estimate Fig. 6.63, on the other hand, recognises the modified viscosity on the wall face, thus giving a different picture of the relative importance of the near-wall error to the rest of the domain.

Figs. 6.65, 6.66 and 6.67 show the estimated error distribution in k around the hill for the Direct Taylor Series, Moment and Residual Error estimate. High errors are detected just downstream from the detachment point, where high shear causes high generation of k . The second region of high error is associated with the reattachment point, where ϵ varies rapidly. The Direct Taylor Series Error estimate (Fig. 6.65), places the error more towards the middle of the vortex as a consequence of the inaccurate evaluation of the second gradient close to the boundary.

Error estimates for ϵ are presented in Figs. 6.68, 6.69 and 6.70. The principle error source is associated with high gradients around and just downstream of the detachment point (see Fig. 6.61) and are highlighted by all methods. The Residual Error estimate also highlights the reattachment region, characterised by the rapidly varying ϵ .

Figs. 6.71, 6.72 and 6.73 show the distribution of the estimated error norm for velocity, turbulent kinetic energy and its dissipation. The high error is again concentrated in several points around the hill summit. Error norms for k and ϵ detect the peak error around the detachment and reattachment points behind the hill.

The complexity of the mathematical model for turbulent flows does not allow us to create a numerical solution on a very fine mesh and treat it as a good approximation of the exact solution. This is partly due to the high gradients in turbulence properties requiring a very fine mesh and partly to the irreducible error introduced by the wall-functions. The applicability of wall-function in terms of the y^+ condition limits the minimum size of the cell near the wall, thus limiting the reduction of the discretisation error. This can be avoided if a low- Re turbulence model is used (eg. Launder and Sharma [76]). The change in the turbulence model, however, also changes the modelling error, as a result of which the two solutions cannot be compared directly. The accuracy of error estimation in the present case will therefore be judged only by comparing the different error estimates and their scaling properties.

The scaling of the estimated mean and maximum error for the velocity is given in

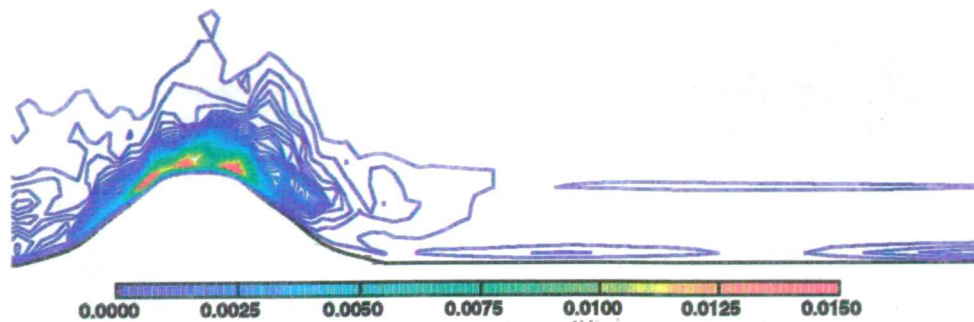


Figure 6.71: Turbulent flow over a 2-D hill: estimated error norm for velocity, uniform mesh, 2044 CV.

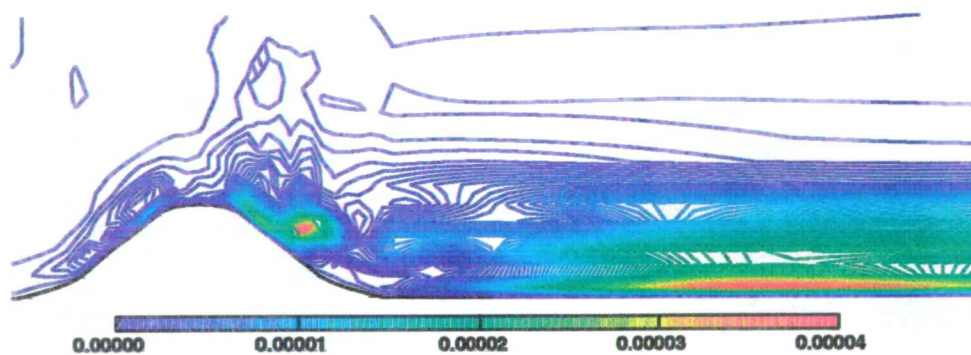


Figure 6.72: Turbulent flow over a 2-D hill: estimated error norm for k , uniform mesh, 2044 CV.

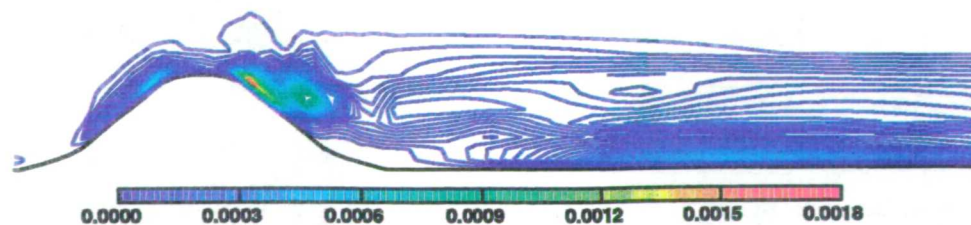


Figure 6.73: Turbulent flow over a 2-D hill: estimated error norm for ϵ , uniform mesh, 2044 CV.

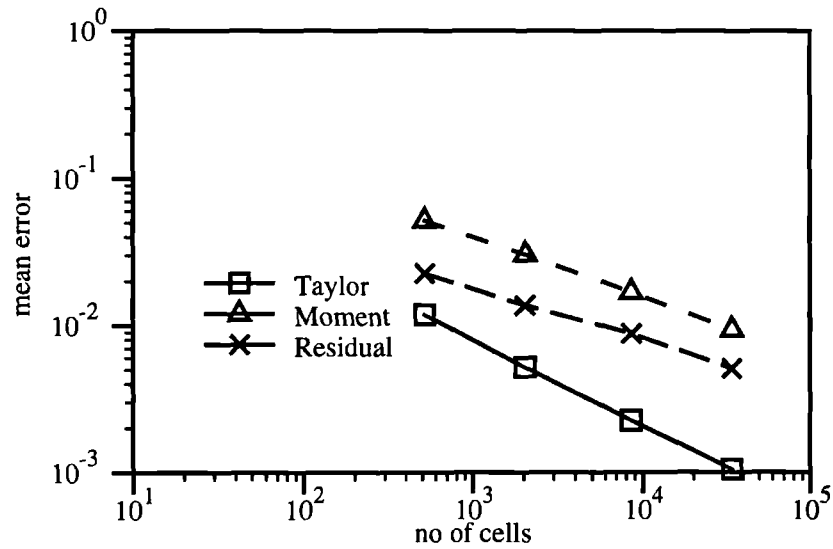


Figure 6.74: Turbulent flow over a 2-D hill: scaling of the mean velocity error for uniform refinement.

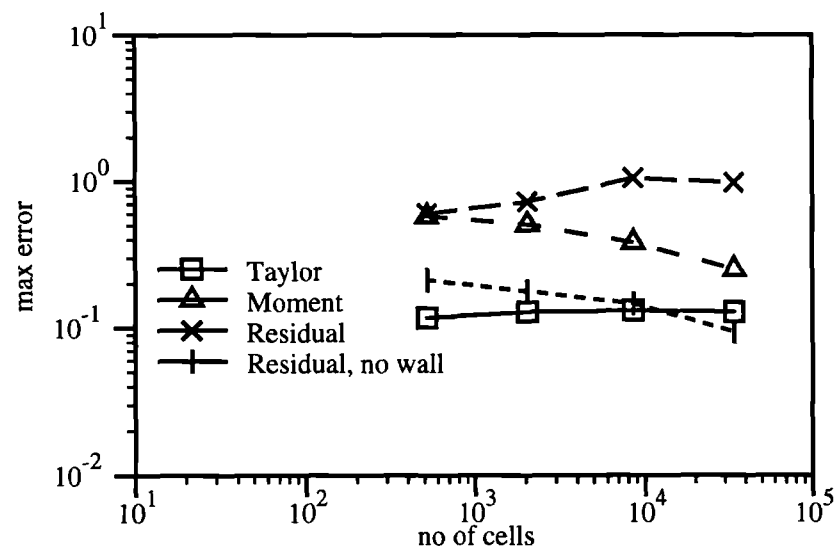


Figure 6.75: Turbulent flow over a 2-D hill: scaling of the maximum velocity error for uniform refinement.

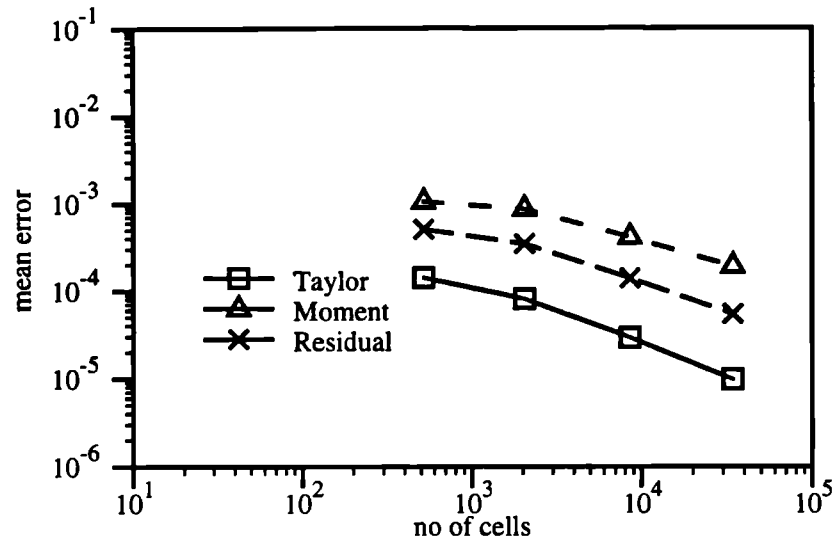


Figure 6.76: Turbulent flow over a 2-D hill: scaling of the mean error in k for uniform refinement.

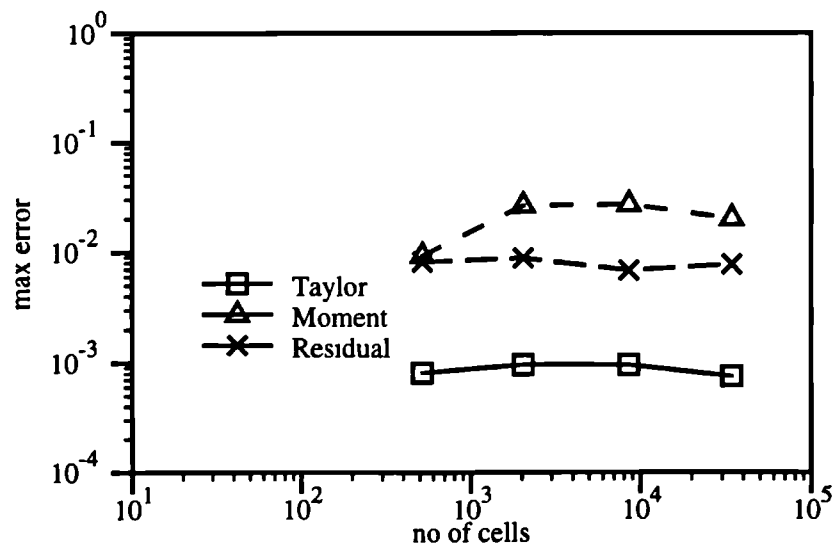


Figure 6.77: Turbulent flow over a 2-D hill: scaling of the maximum error in k for uniform refinement.

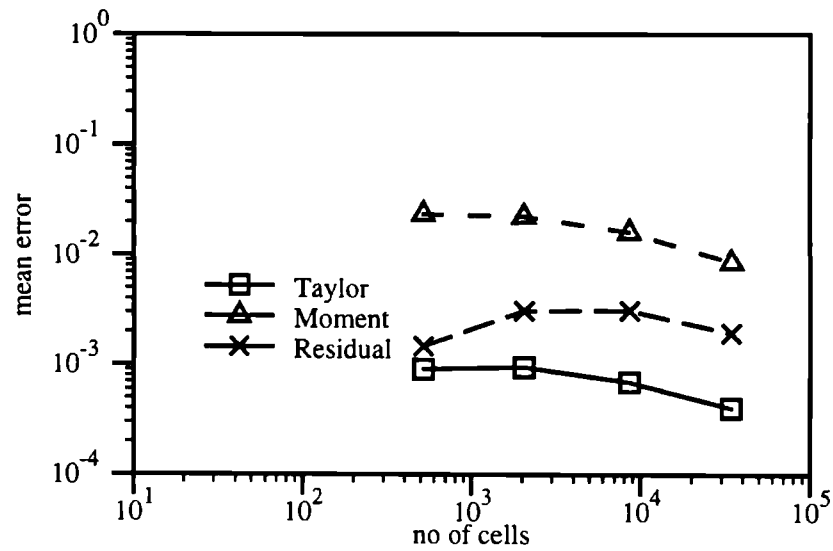


Figure 6.78: Turbulent flow over a 2-D hill: scaling of the mean error in ϵ for uniform refinement.

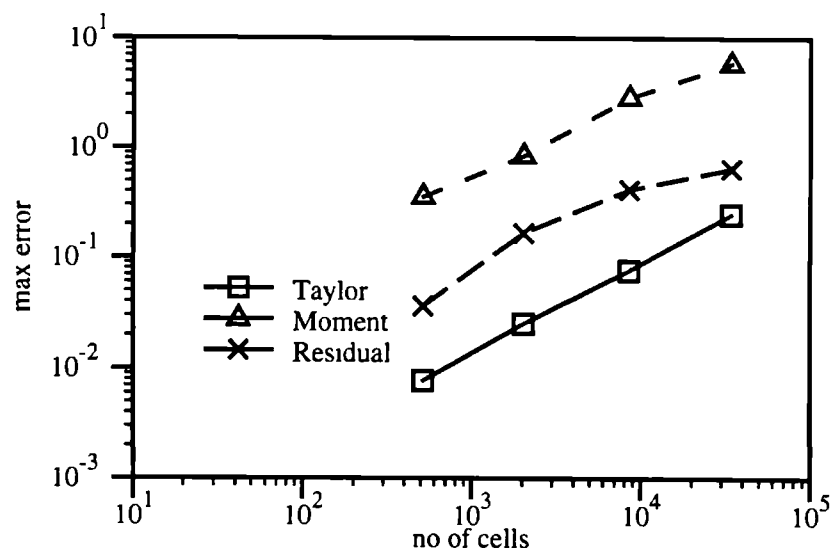


Figure 6.79: Turbulent flow over a 2-D hill: scaling of the maximum error in ϵ for uniform refinement.

Figs. 6.74 and 6.75. If it is assumed that the Moment and Residual Error estimates bound the exact error, as was previously the case, the mean velocity error for the mesh with 2044 CV-s can be estimated to 0.02 m/s or 1 % of the centreline velocity. Fig. 6.75 contains two graphs for the Residual Error estimate, with and without the values at the near-wall cells. If the near-wall cells are excluded, the Residual and Moment Error estimates follow the same path. The Direct Taylor Series Error estimate shows a peculiarity: as the number of cells increases, the maximum error also increases, bringing the error estimate into close agreement with the other two methods. As mentioned earlier, this is a consequence of the more accurate evaluation of the second velocity gradient. After 3 levels of uniform refinement the maximum error has been reduced only by about 50 %. Further refinement decreases the near-wall y^+ below the recommended range, causing higher modelling errors. A similar behaviour can be seen in the scaling of estimated errors in turbulent kinetic energy, Figs. 6.76 and 6.77. The maximum error remains virtually unchanged through the whole refinement procedure. The Moment and Residual Error estimates are very close to each other, both in the mean and maximum value. The Direct Taylor Series Error estimates predicts lower error levels.

The change of the mean and maximum error in ϵ shown in Figs. 6.78 and 6.79 gives a very different picture of the solution accuracy. Not only does the reduction of the mean error appear to be very slow, but the maximum error actually increases with refinement. It should be noted that the near-wall cell has been excluded from all error estimates, consistent with the wall-function treatment. Fig. 6.61 shows the high local peak in ϵ at the point of detachment on the lee slope. On the mesh with 2044 CV-s, this peak is resolved over a single cell. As the mesh becomes finer, the magnitude of ϵ increases. Even on the finest mesh, the peak in ϵ is only one cell wide. As a result of this behaviour, the maximum error (Fig. 6.79) actually increases with mesh refinement.

It can be concluded that the wall-function treatment carries an irreducible error caused in two ways. Firstly, the details of the near-wall flow are not resolved unless the y^+ condition is violated. If that is the case, wall-functions do not enforce

the logarithmic near-wall velocity profile and high- Re turbulence models produce incorrect behaviour of the turbulence properties in the low- Re region near the wall, resulting in high modelling errors (see *eg.* Patel *et al.* [108]). If, on the other hand, the y^+ condition is obeyed, it limits the cell size near the wall, thus fixing the level of the discretisation error.

In the adaptive calculations, the combination of the Moment Error estimate for the velocity and the Residual Error estimate for k has been used. The y^+ values in near-wall cells are not allowed to drop below 50, by explicitly excluding the cells from refinement parallel with the wall. It is, however, still allowed to split the near-wall cells in a way that does not violate the y^+ condition. The laminar adaptive calculation implies that the major factor influencing the mesh quality is the unrefinement procedure. Having in mind that the typical error distribution in turbulent calculations contains high local error peaks, the refinement/unrefinement parameters for this calculations have been set to:

$$\lambda_{ref} = 1.8$$

$$\lambda_{unref} = 0.2$$

$$\xi = 0.6$$

The first set of results is obtained by refining the initial uniform mesh with 2044 CV-s. The mesh changes in the first three levels of refinement are shown in Figs. 6.80, 6.81 and 6.82. The changes in the Moment Error estimate for velocity through the levels of refinement can be followed in Figs. 6.83, 6.84 and 6.85, with the estimated error in k in Figs. 6.86, 6.87 and 6.88. The error reduction shows a very different behaviour than in the case of laminar flow. The first level of refinement reduces the error magnitude, but the maximum error remains at the same place, just downstream from the summit. In the second level of refinement, the error peak is moved to the near-wall cell, with only a slight change in its magnitude. This cell is consequently blocked from further refinement on the basis of the y^+ condition. Further adaptive mesh changes are unable to cause any change in the maximum error.

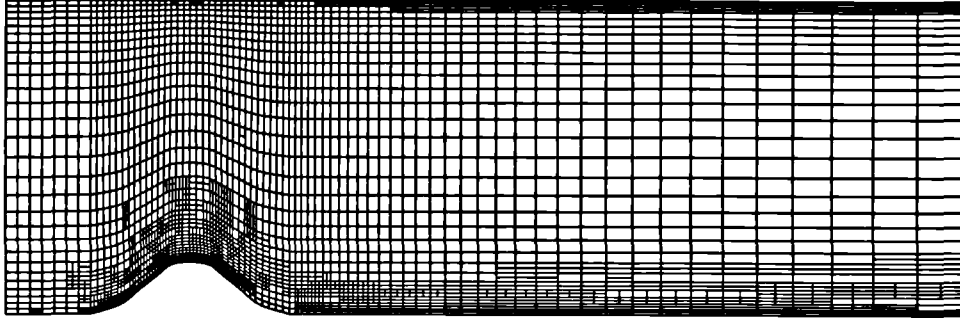


Figure 6.80: Turbulent flow over a 2-D hill: mesh after the first level of refinement.

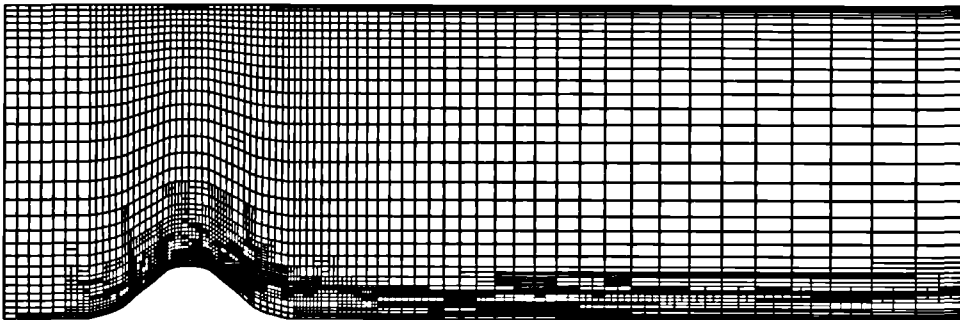


Figure 6.81: Turbulent flow over a 2-D hill: mesh after the second level of refinement.

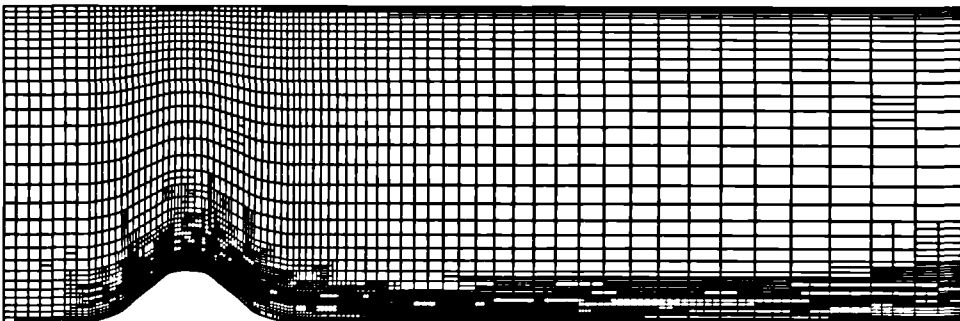


Figure 6.82: Turbulent flow over a 2 D hill: mesh after the third level of refinement.

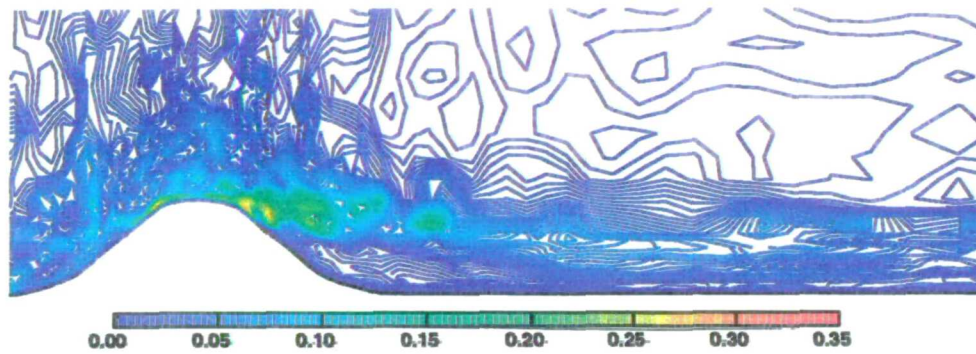


Figure 6.83: Turbulent flow over a 2-D hill: Moment Error estimate for velocity after the first level of adaptive refinement.

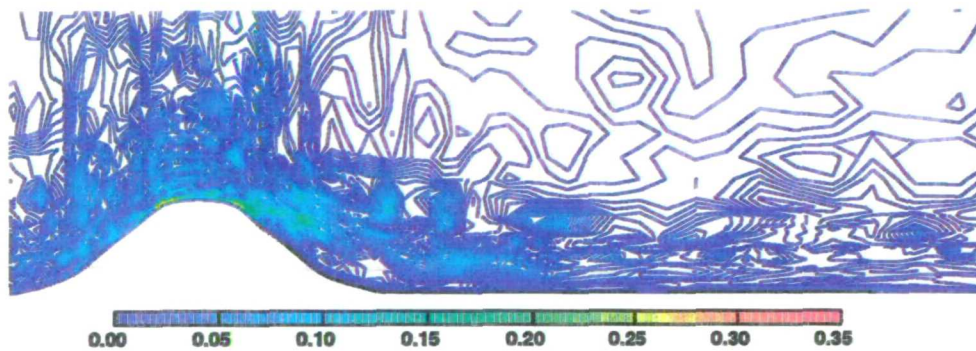


Figure 6.84: Turbulent flow over a 2-D hill: Moment Error estimate for velocity after the second level of adaptive refinement.

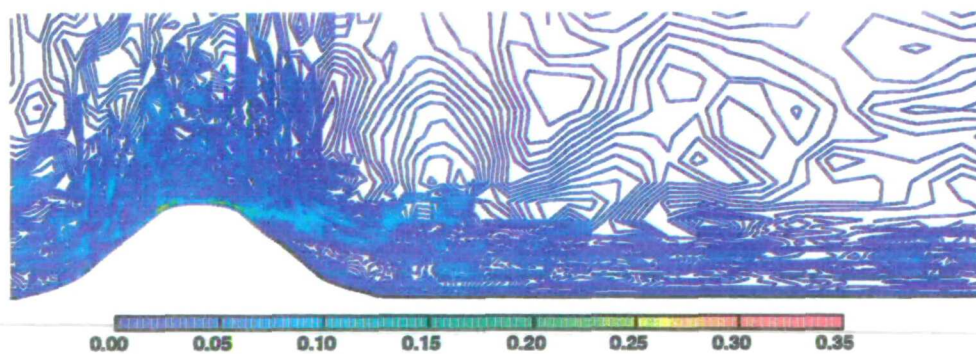


Figure 6.85: Turbulent flow over a 2-D hill: Moment Error for velocity estimate after the third level of adaptive refinement.

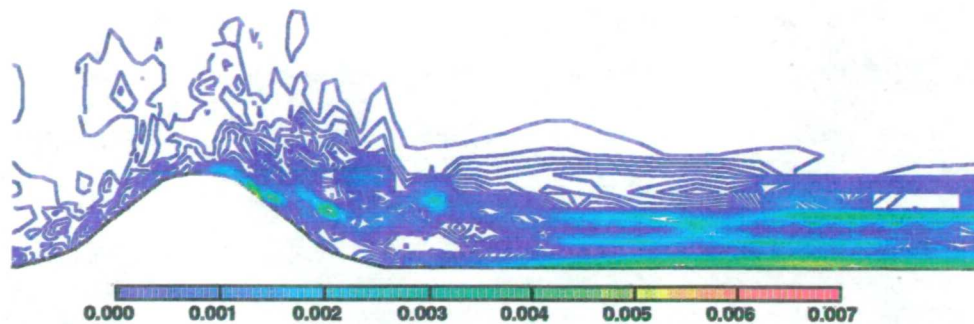


Figure 6.86: Turbulent flow over a 2-D hill: Residual Error estimate after the first level of adaptive refinement.

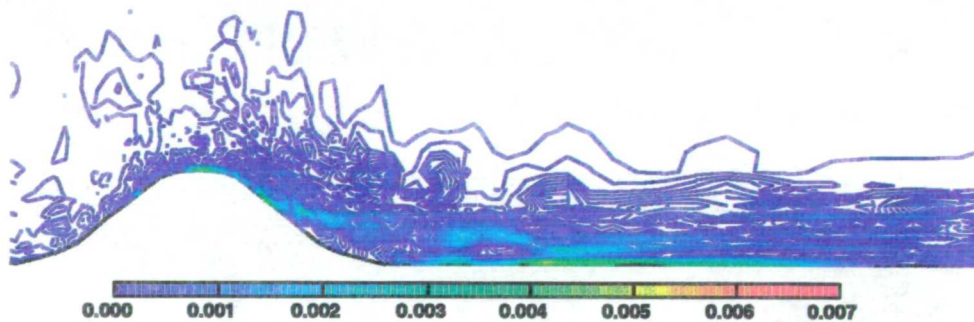


Figure 6.87: Turbulent flow over a 2-D hill: Residual Error estimate after the second level of adaptive refinement.

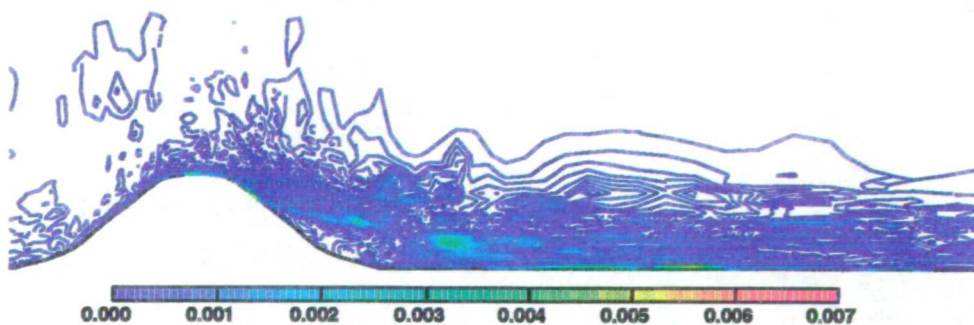


Figure 6.88: Turbulent flow over a 2-D hill: Residual Error estimate after the third level of adaptive refinement.

The error reduction for the turbulent kinetic energy is somewhat more favourable, as it is initially possible to improve the mesh resolution without trapping the maximum error near the wall. The initial region of high error is located around the reattachment point (see Fig. 6.67) and is removed with the additional resolution. In the third refinement level, Fig. 6.88, the maximum error is eventually trapped in the near-wall cell on the lee slope of the hill and cannot be further reduced.

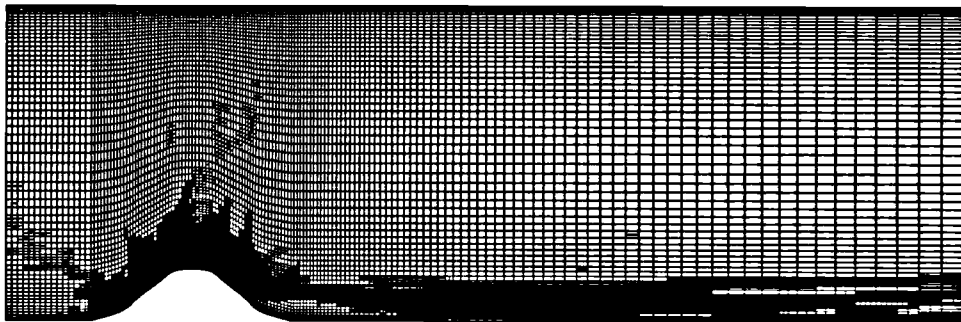


Figure 6.89: Turbulent flow over a 2-D hill: second level of refinement-only from the fine mesh.

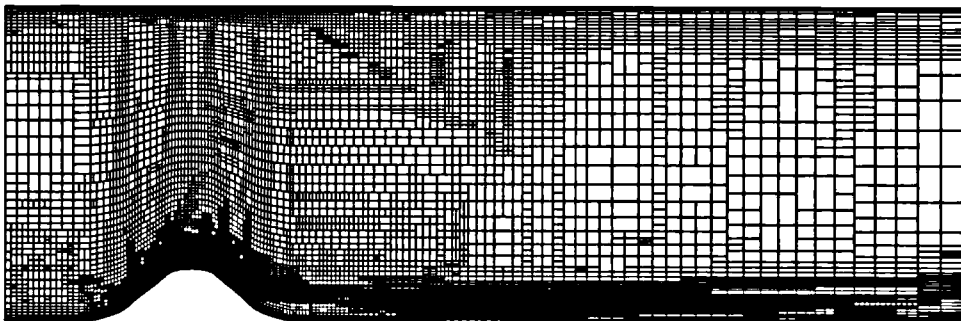


Figure 6.90: Turbulent flow over a 2-D hill: second level of refinement/unrefinement from the fine mesh.

It is also interesting to follow the behaviour of the local peak in the distribution of ϵ through mesh refinement. The coarse mesh shows the high ϵ in the near-wall cell. As the refinement progresses, the peak remains in the near-wall cell, rapidly increasing in magnitude. Even after four levels of refinements it is still resolved over only a single cell, raising doubts about the interaction between the wall-functions and the formulation of the ϵ equation used in the turbulence model. The gradient of ϵ therefore rises as the mesh resolution improves, causing an increase in the maximum

error similar to the one in the case of uniform refinement (see Fig. 6.79).

The refinement-only and refinement/unrefinement calculations for this test case have been performed starting from the uniform mesh with 8584 CV-s. Turbulent flows show a much larger difference between the mean and maximum error than was the case in laminar flows, thus opening a potential for the use of unrefinement. Figs. 6.89 and 6.90 show the meshes obtained after two levels of refinement-only and refinement/unrefinement. The additional cells are again concentrated around the summit of the hill and in the recirculation region. In the middle of the duct, the profiles of velocity, k and ϵ are flat, the local error is low and the unrefinement can successfully remove the unnecessary cells. The lower quality of the unrefined mesh does not play an important role: the peak error is still associated with the very thin boundary layers, particularly around the summit of the hill. In spite of the fact that the initial mesh was graded towards the wall, the refinement procedure tends to introduce considerably more resolution than was originally the case.

One of the unwritten rules of the “manual” mesh generation for turbulent flows is that the long and thin cells are appropriate close to the wall, as they provide sufficiently good resolution for the high velocity gradients normal to the wall and do not impose unnecessary resolution along the wall². This approach is only partially appropriate for detached flows: around the detachment and reattachment points, the velocity gradient is high not only normal to the wall, but also parallel to it. For good accuracy, it is necessary to provide sufficient resolution in both directions. The adaptive procedure satisfies this condition by splitting the affected cells into four. If the flow is attached, the directionality of the mesh is preserved, thus preventing unnecessary overheads.

The Moment Error estimate for the velocity and the Residual Error estimate for k , Figs. 6.91 and 6.92, show a similar distribution as in the case of refinement-only from the coarse initial mesh. The peak error is again trapped in the near-wall cell, out of reach of the adaptive refinement procedure. The estimated error levels are slightly higher, but the irreducible error from the wall-functions still exists.

²Such an approach has been used to create the initial mesh for the hill, shown in Fig. 6.31.

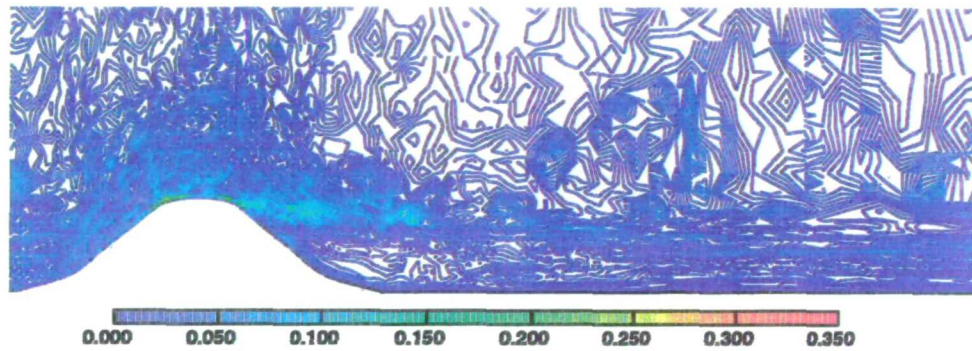


Figure 6.91: Turbulent flow over a 2-D hill: Moment Error estimate for velocity after two levels of refinement/unrefinement.

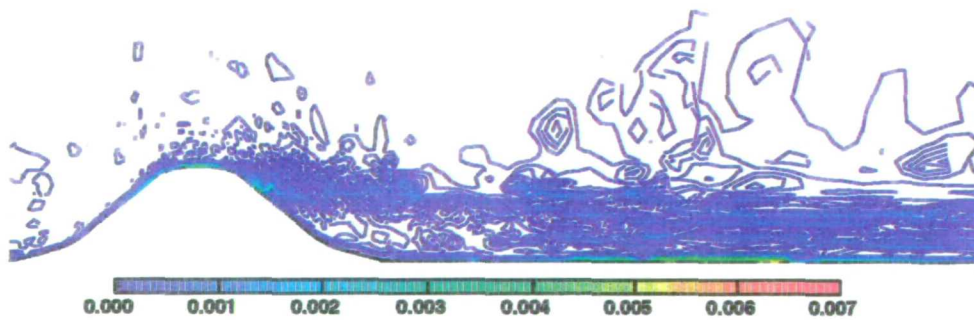


Figure 6.92: Turbulent flow over a 2-D hill: Residual Error estimate for k after two levels of refinement/unrefinement.

Let us finally examine the error reduction rate for the different methods of adaptive refinement. The scaling of the maximum value for the Moment Error estimate for velocity, with and without the near-wall cells is given in Figs. 6.93 and 6.94. If the near-wall cells are included, the error in velocity actually increases with mesh refinement. The high resolution of the adaptively refined mesh picks up the rapid variation of the velocity around the detachment point, but the error cannot be further reduced because of the y^+ condition. When the near-wall cells are excluded (both in uniform and adaptive refinement), the adaptive calculation shows its superiority. The error reduction is faster if the refinement starts from the fine initial mesh. The benefit of unrefinement can also be seen: the refinement/unrefinement graph moves the points in the refinement-only curve towards the lower number of cells.

The Moment and Residual Error estimates for k are shown in Figs. 6.95 and 6.96. The Moment Error estimate shows a very slight error reduction with the refinement, whereas the maximum Residual Error increases, bringing the two estimates into close agreement.

In terms of the maximum error reduction, the adaptive algorithm shows no significant advantage over uniform refinement. The apparent increase in the error is a consequence of the finer mesh resolution normal to the wall, especially around the detachment point. It should also be mentioned that the final level of uniform refinement (34336 CV-s) violates the y^+ condition on some parts of the lower wall, causing a change in the wall-function treatment from the log-law to the laminar sublayer regime. The discretisation error therefore seems to be somewhat smaller, but a modelling error has been introduced instead.

The performance of the adaptive procedure can also be measured in terms of its influence on the solution accuracy. For this purpose, two uniform mesh solutions (2044 and 34336 CV-s) will be compared with the results of the adaptive procedure starting from the coarse initial mesh (2044 CV). The presented results show the distribution of the x -component of the velocity and the turbulent kinetic energy at $x = 0$ and $x = 0.134\text{ m}$.

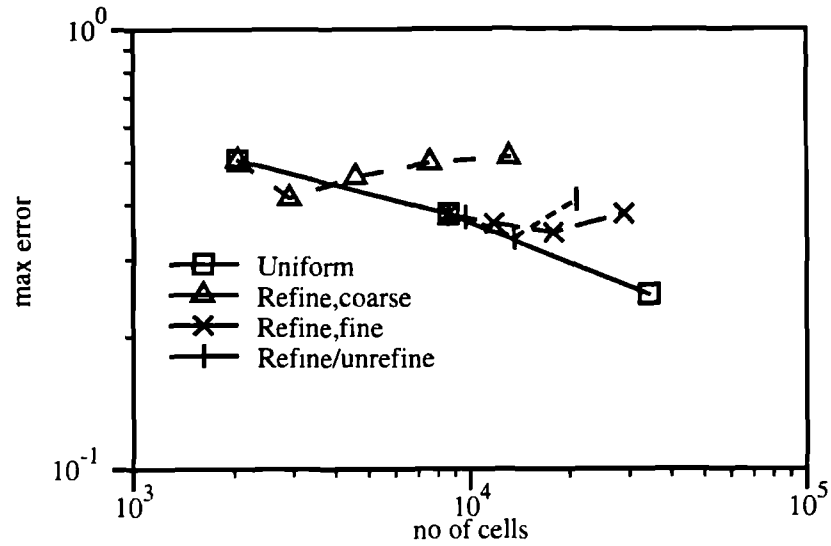


Figure 6.93: Turbulent flow over a 2-D hill: scaling of the maximum Moment Error for velocity for different types of refinement.

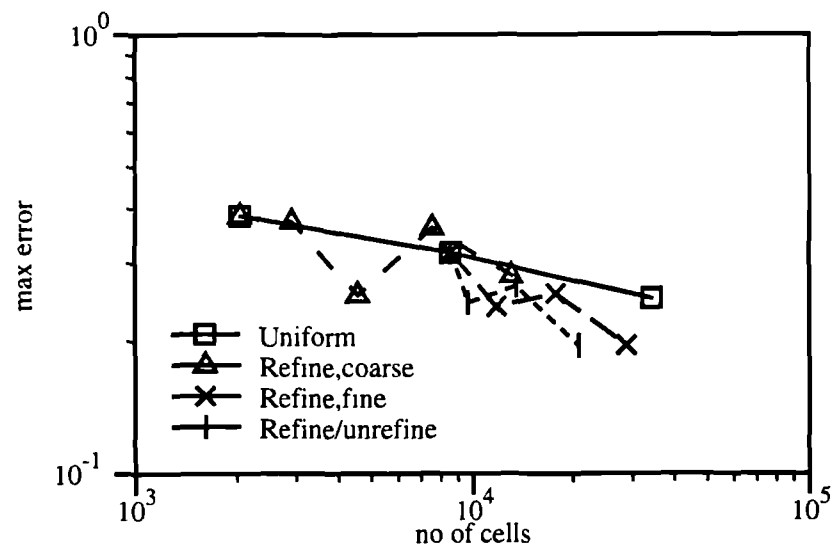


Figure 6.94: Turbulent flow over a 2-D hill: scaling of the maximum Moment Error for velocity without the near-wall cells for different types of refinement.

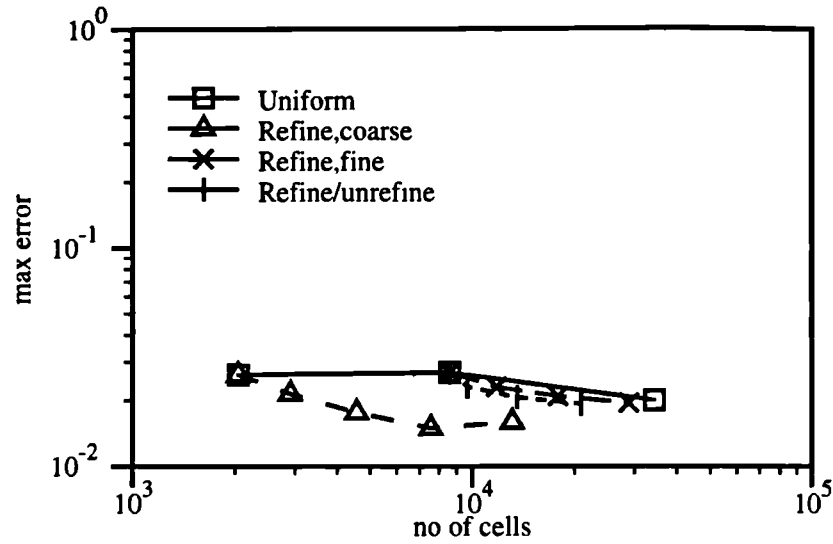


Figure 6.95: Turbulent flow over a 2-D hill: scaling of the maximum Moment Error for k for different types of refinement.

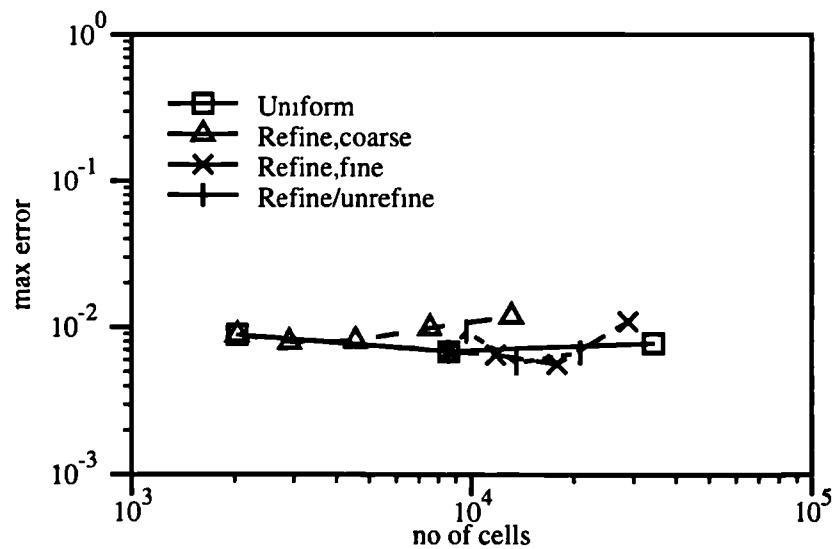


Figure 6.96: Turbulent flow over a 2-D hill: scaling of the maximum Residual Error for k for different types of refinement.

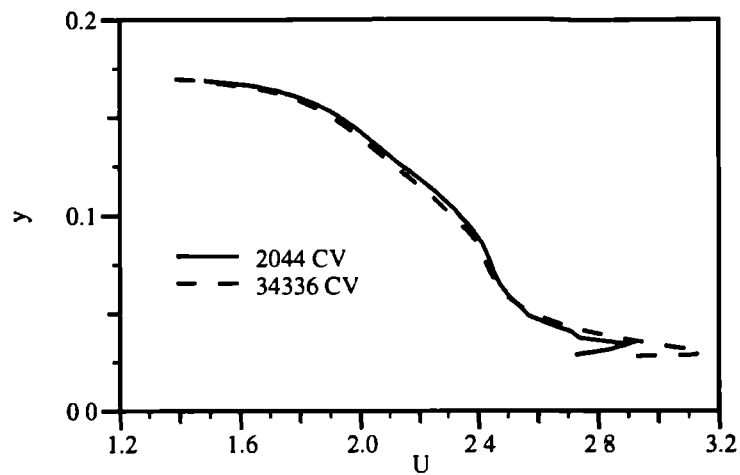


Figure 6.97: Turbulent flow over a 2-D hill: velocity distribution at $x = 0$ for uniform meshes.

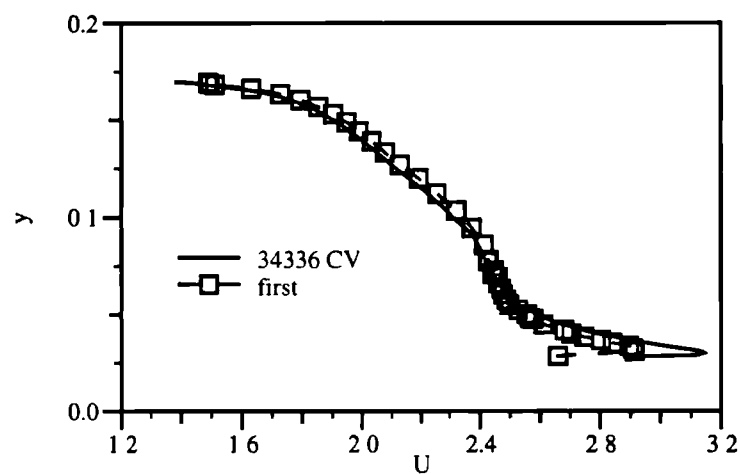


Figure 6.98: Turbulent flow over a 2-D hill: velocity distribution at $x = 0$, first level of refinement.

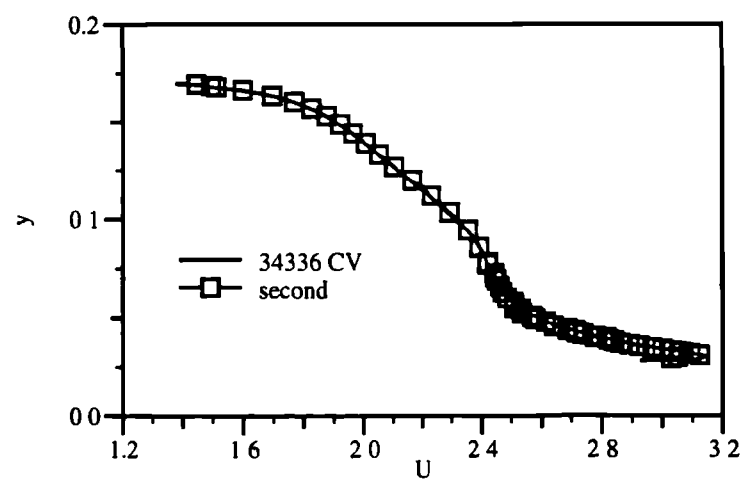


Figure 6.99: Turbulent flow over a 2-D hill: velocity distribution at $x = 0$, second level of refinement.

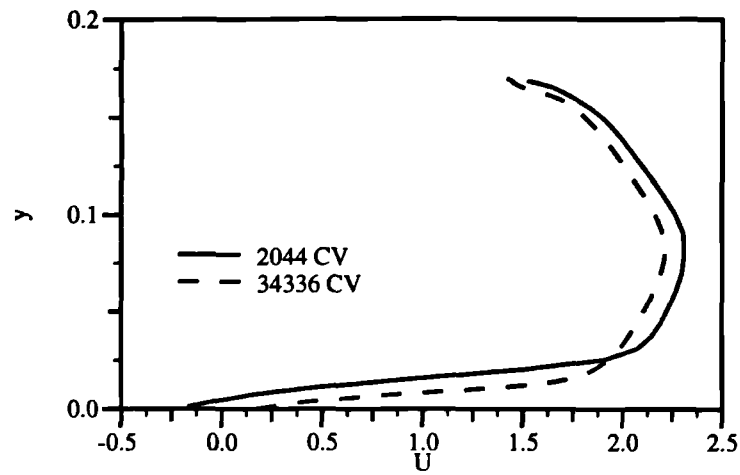


Figure 6.100: Turbulent flow over a 2-D hill: velocity distribution at $x = 134$ for uniform meshes.

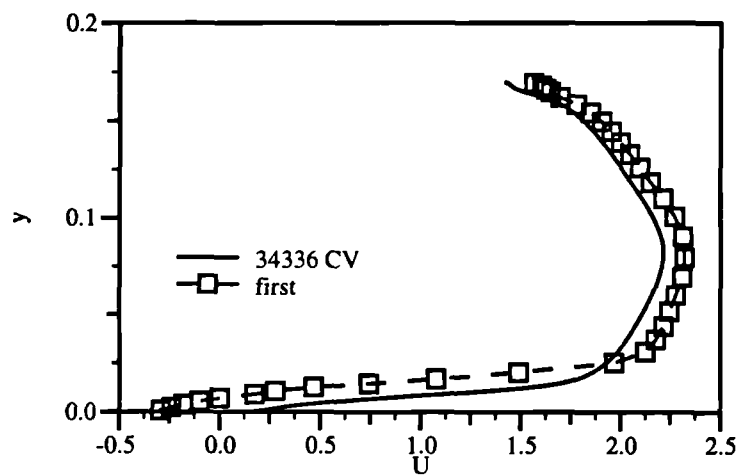


Figure 6.101: Turbulent flow over a 2-D hill: velocity distribution at $x = 134$, first level of refinement.

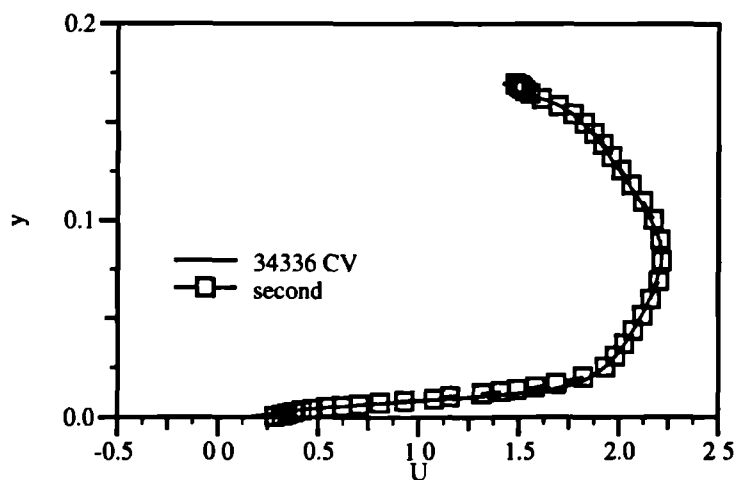


Figure 6.102: Turbulent flow over a 2-D hill: velocity distribution at $x = 134$, second level of refinement.

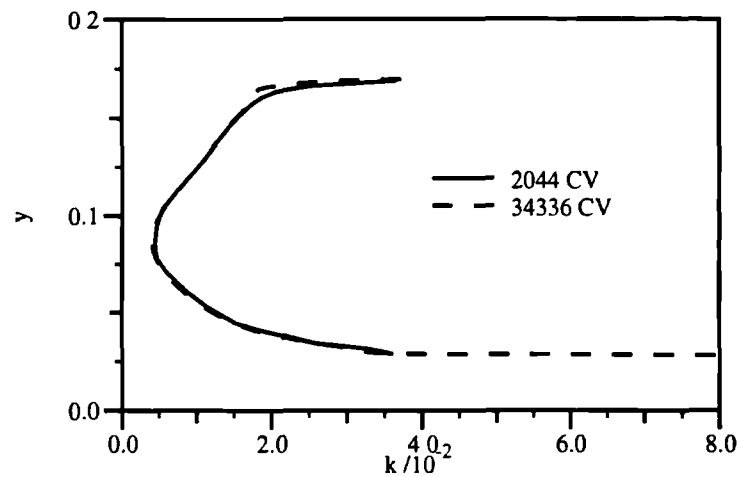


Figure 6.103: Turbulent flow over a 2-D hill: distribution of k at $x = 0$ for uniform meshes.

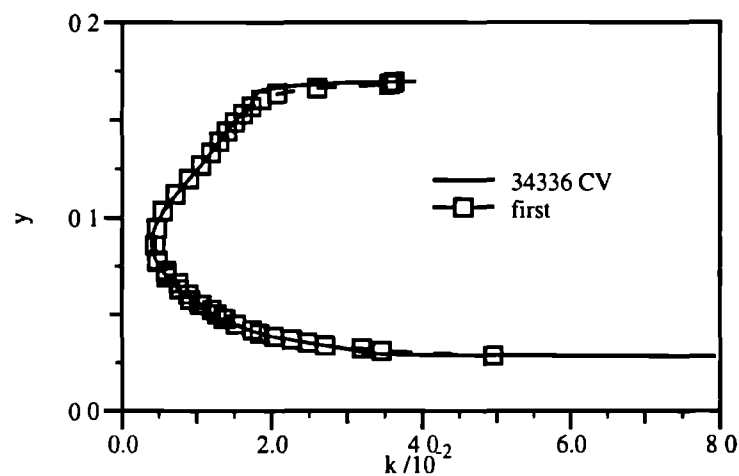


Figure 6.104: Turbulent flow over a 2-D hill: distribution of k at $x = 0$, first level of refinement.

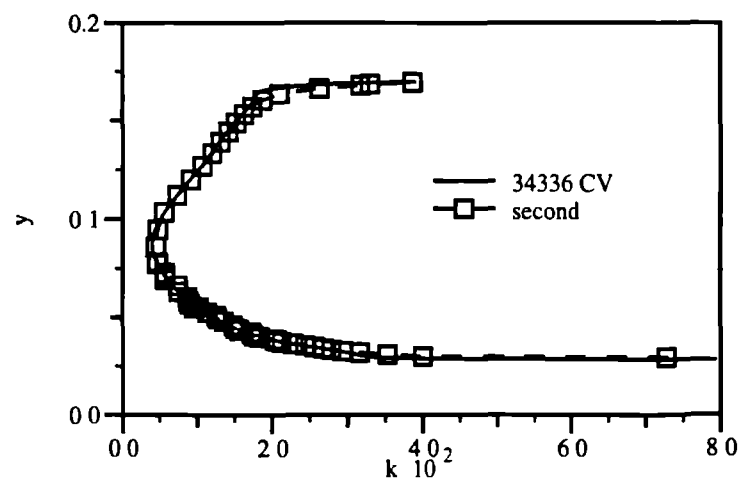


Figure 6.105: Turbulent flow over a 2-D hill: distribution of k at $x = 0$, second level of refinement.

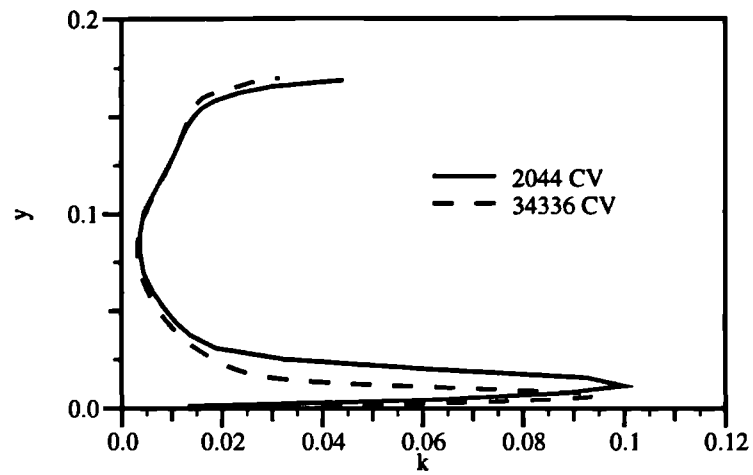


Figure 6.106: Turbulent flow over a 2-D hill: distribution of k at $x = 134$ for uniform meshes.

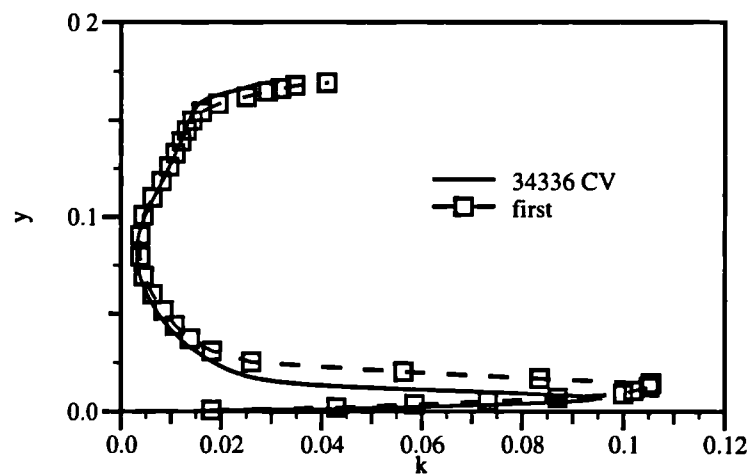


Figure 6.107: Turbulent flow over a 2-D hill: distribution of k at $x = 134$, first level of refinement.

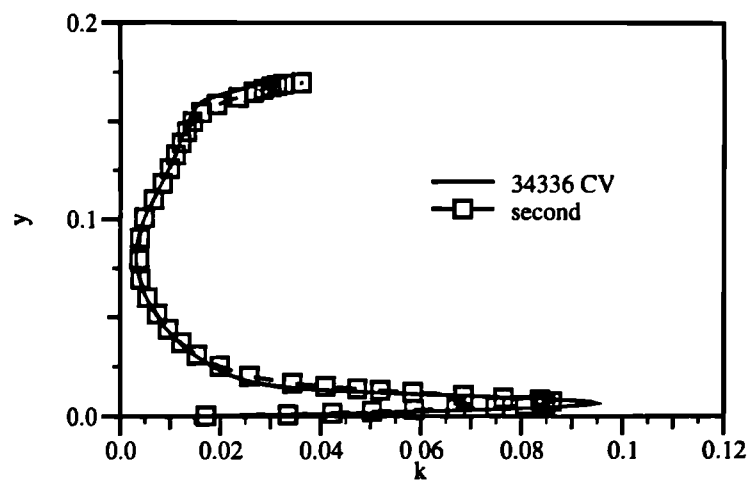


Figure 6.108: Turbulent flow over a 2-D hill: distribution of k at $x = 134$, second level of refinement.

For reference, the difference between the two velocity fields from the uniform meshes is given in Figs. 6.97 and 6.100. The influence of the mesh resolution is clearly visible in both graphs, particularly close to the lower wall. It can also be seen that the flow on the finer mesh is already attached at $x = 0.134\text{ m}$ (Fig. 6.100), whereas the coarse mesh solution still predicts flow recirculation. Figs. 6.98, 6.99, 6.101 and 6.102 follow the development of the solution in the first two levels of refinement for both locations. The positions of cell centres for the adaptively refined meshes are marked with squares. After the first refinement level, the refined mesh consists of 2910 CV-s, with the second level of refinement increasing the number of cells to 4564. It can be seen that the adaptive solution becomes closer to the fine mesh solution in the first level of refinement, with the difference between the two becoming negligible after only two refinement levels.

The distribution of k on the selected locations for two uniform meshes is given in Figs. 6.103 and 6.106. The differences are again considerable, particularly at $x = 0.134\text{ m}$. The adaptive procedure, Figs. 6.104, 6.105, 6.107 and 6.108, again rapidly improves the quality of the solution. The resolution of the sharp profile of k at $x = 0.134\text{ m}$ (Figs. 6.107 and 6.108) seems to have a very strong influence on the length of the recirculation bubble.

The above example clearly shows that the proposed local refinement algorithm makes it possible to create a solution which is for all practical purposes as accurate as the fine mesh solution on the mesh with a much smaller number of cells.

The alternative to the troublesome wall-functions in the treatment of the wall is the integration all the way to the wall with the use of a low-Reynolds-number turbulence model. A wide variety of extensions for the $k - \epsilon$ family of two-equation models exists in the literature (*eg.* Lam and Bremhorst [74], Launder and Sharma [76], Chien, [29] *etc.*). These models reproduce the correct near-wall behaviour of k and ϵ and no longer limit the refinement close to the wall. The application of a low- Re model, however, introduces a different set of problems. If the rapid variation of k and ϵ near the wall is to be resolved, the mesh in that region needs to be very fine indeed, with the first cell falling into the region of $y^+ \sim 0.1$ (Leschziner [86]). An

example of such a mesh around the hill is shown in Fig. 6.109. This mesh has been used in conjunction with the low- Re turbulence model by Launder and Sharma [76] to produce the velocity field shown in Fig. 6.110. A closer look at the velocity field close to the hill reveals the difficulty: on the windward slope, the boundary layer is compressed so hard that even this mesh cannot accurately resolve the high velocity gradient. Instead of reducing the error, the attempt to resolve the boundary layer close to the summit of the hill makes the problem more difficult. All error estimates recognise this point as the highest error source. As the wall-functions are not in use, the error estimates can be used without any modification. As the boundary layer thickness increases, the low- Re approach resolves the velocity profile more accurately. The accuracy of the solution in terms of modelling errors is considered to be better, particularly around flow detachment and reattachment.

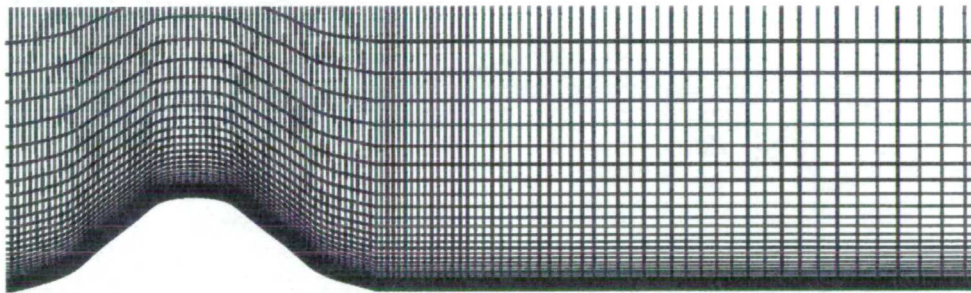


Figure 6.109: Turbulent flow over a 2-D hill: uniform mesh for the low- Re calculation.

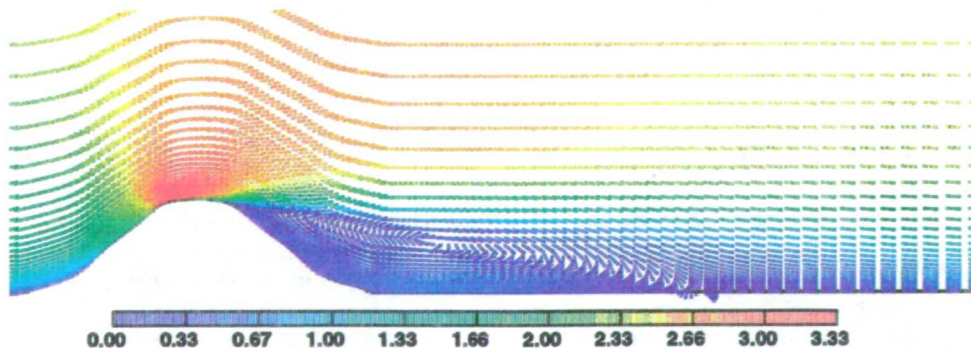


Figure 6.110: Turbulent flow over a 2-D hill: velocity field for the low- Re turbulence model.

If it were possible to decrease the high error peak close to the summit of the hill, the cost associated with the use of the low- Re model could be considered acceptable. For this purpose, the refinement-only procedure starting from the mesh shown in Fig. 6.109 is used. The initial mesh consists of 13320 CV-s with strong grading towards the wall. The y^+ value in the near-wall cell is of the order of 0.1. The method of error estimation and the refinement parameters are kept the same as in previous calculations. After three levels of refinement, the mesh consists of 38076 CV-s. According to the error estimates, even this mesh is too coarse to resolve the compressed boundary layer at the windward side of the hill. Figs. 6.111, 6.112 and 6.113 show the change in the estimated maximum error with refinement. The be-

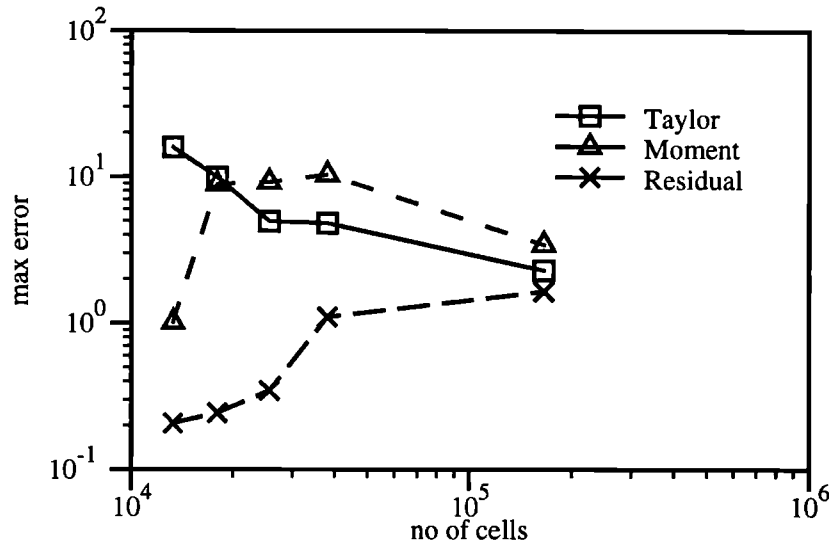


Figure 6.111: Turbulent flow over a 2 D hill: maximum error reduction for velocity with adaptive refinement and the low- Re turbulence model.

haviour of the estimates is slightly different than before. For the maximum velocity error, the Moment and Direct Taylor Series Error estimates agree closely and show that the error levels off after two refinement levels. The mesh is now fine enough to start resolving the strongly compressed boundary layer close to the summit of the hill. The Residual Error estimate still shows an increase in the error, but comes to a closer agreement with other two error estimates. Only the final level of refinement actually decreases the error. This is, however associated with a large increase in the

number of cells. The majority of cells added in the adaptive procedure are placed in the near-wall region. The “1-irregularity” condition imposed to preserve the mesh quality pushes the number of cells in the final refinement level up to 165470, considerably increasing the cost of the calculation. The result is still impressive: all error estimates on all fields report a drop in the maximum error.

The changes of the maximum error in k and ϵ are somewhat more favourable. After an initial jump in the first level of refinement, the error consistently decreases. The decrease is not uniform, as the position of the error peak changes as the resolution around its previous position becomes better.

This test case allows us to draw several conclusions about the compatibility between local mesh refinement and turbulent flow calculations. The performance of the adaptive algorithm in terms of error reduction is worse than in the case of laminar flows, primarily as a result of the more complicated mathematical model. The major source of difficulties is near the wall, where all fields vary very rapidly. If this region is effectively “bridged” by the wall-functions, the limits on their applicability in terms of modelling errors introduce a limitation on the minimum cell size. When the adaptive procedure moves the maximum error into the near-wall cell, it cannot be reduced any further. Mesh adaptivity, however, does improve the accuracy away from the wall and offers better resolution normal to the wall, which is of particular importance around the detachment and reattachment point.

An alternative wall treatment in turbulent calculations is the use of a low- Re turbulence model with integration all the way down to the wall. This approach requires much higher near-wall resolution, with the first cell falling well into the laminar sublayer. A low- Re model does not limit the adaptive procedure in any way, thus allowing a considerable improvement in accuracy. It should, however, be kept in mind that the rapid near-wall variation requires a very fine mesh and even with an adaptive procedure the necessary number of cells will be much higher than for the wall-function treatment. The peak error still remains near the wall. A potential for better near-wall accuracy can be possibly found in alternative formulations of the two-equation turbulence models, with the turbulence variables that vary linearly

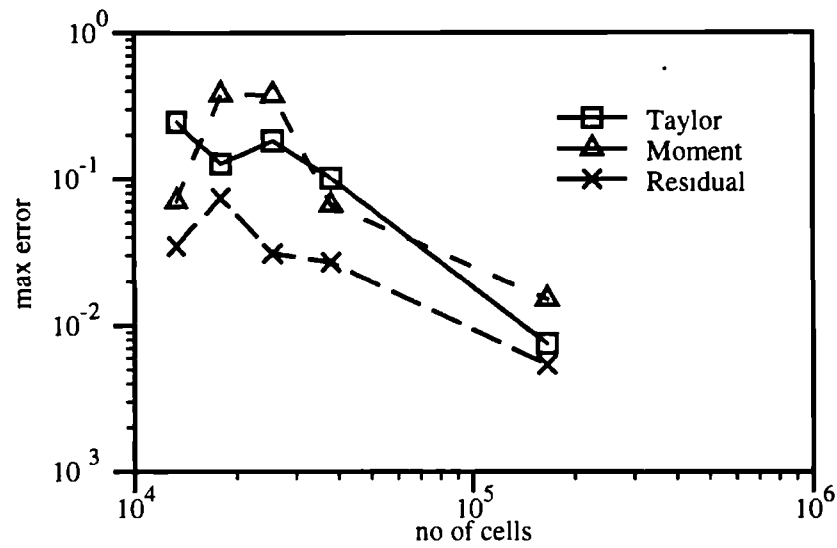


Figure 6.112: Turbulent flow over a 2-D hill: maximum error reduction for k with adaptive refinement and the low- Re turbulence model.

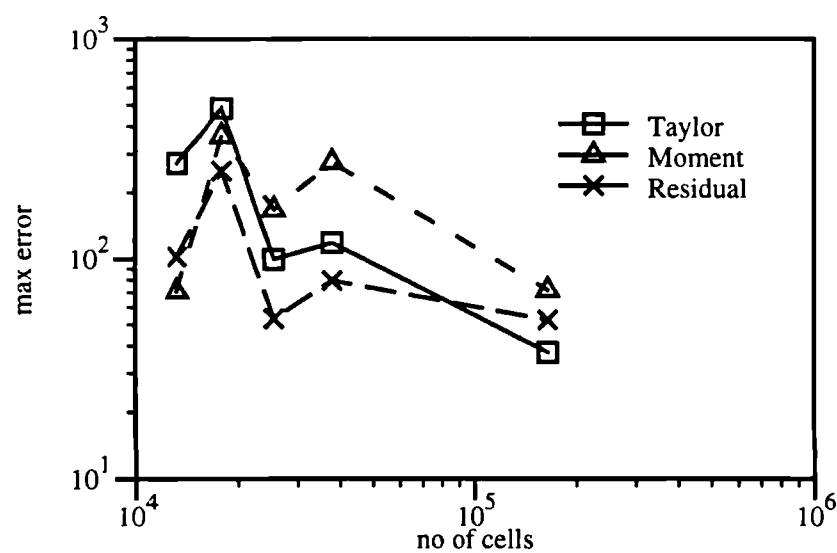


Figure 6.113: Turbulent flow over a 2-D hill: maximum error reduction for ϵ with adaptive refinement and the low- Re turbulence model.

in near the wall, such as the $q - \zeta$ two equation model by Gibson and Dafa'Alla [51, 34].

6.4 Turbulent Flow over a 3-D Swept Backward-Facing Step

The swept backward-facing step test case demonstrates the performance of the local mesh refinement algorithm on a three-dimensional situation. The setup of the case

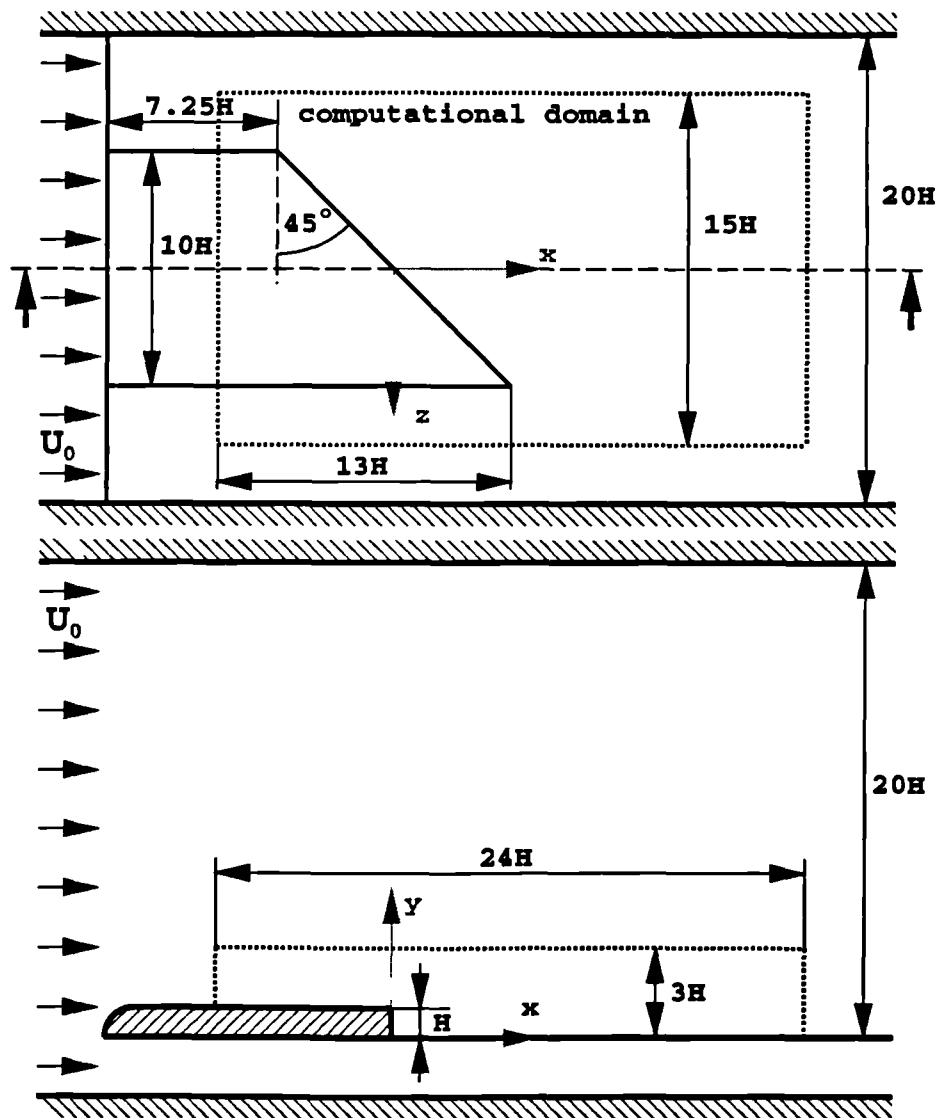


Figure 6.114: 3-D swept backward facing step: test setup.

consists of an obstacle with a blunt leading edge and a triangular back mounted on a plate inside the channel, Fig. 6.114.

The height of the step is $H = 0.04\text{ m}$. The flow in this geometry has been experimentally investigated by LeHuuNho and Béguier [80], using visualisation and Laser-Doppler Anemometry (LDA) in a water channel at $Re = 3000$ and a rotating hot-wire probe in the wind-tunnel at $Re = 60000$. At high Reynolds number, a vortex is formed behind the upstream corner of the step from the rolling of separating shear layers (Moinat and Lesieur [96]). The vortex follows the step and is eventually straightened by the side stream at the other side of the step.

Calculations for the same geometry on $Re = 60000$ have been done by Moinat and Lesieur [96] using Large Eddy Simulation (LES) with the Structure function sub-grid scale turbulence model (Métais and Lesieur [94]) and the standard $k - \epsilon$ model with wall-functions.

In order to provide a reasonable mesh resolution around the vortex, the computational domain covers only a small region around and behind the step, marked in Fig. 6.114. The inlet boundary is positioned downstream of the leading edge, thus avoiding the modelling problems related to the laminar separation and transition to turbulence. Unfortunately, no experimental data is available upstream of the triangular part of the obstacle. Following Moinat and Lesieur [96], the velocity distribution at the inlet is prescribed as a $1/7$ power-law profile with a boundary layer thickness of $\delta = 0.3 H$ in the x -direction. The other two velocity components are set to zero. On the top surface and the sides of the computational domain, a symmetry plane boundary condition is specified. The free-stream velocity is set to 22.5 m/s , corresponding to the measured wind-tunnel conditions.

The turbulence model selected for this calculation is again the non-linear version of the $k - \epsilon$ model by Shih [123] with wall-functions. The turbulent kinetic energy and its dissipation at the inlet are prescribed to be in equilibrium with the velocity field.

The flow field is particularly complex around the corner of the triangular part, where the vortex is created. Previous calculations suggest that the mesh required to

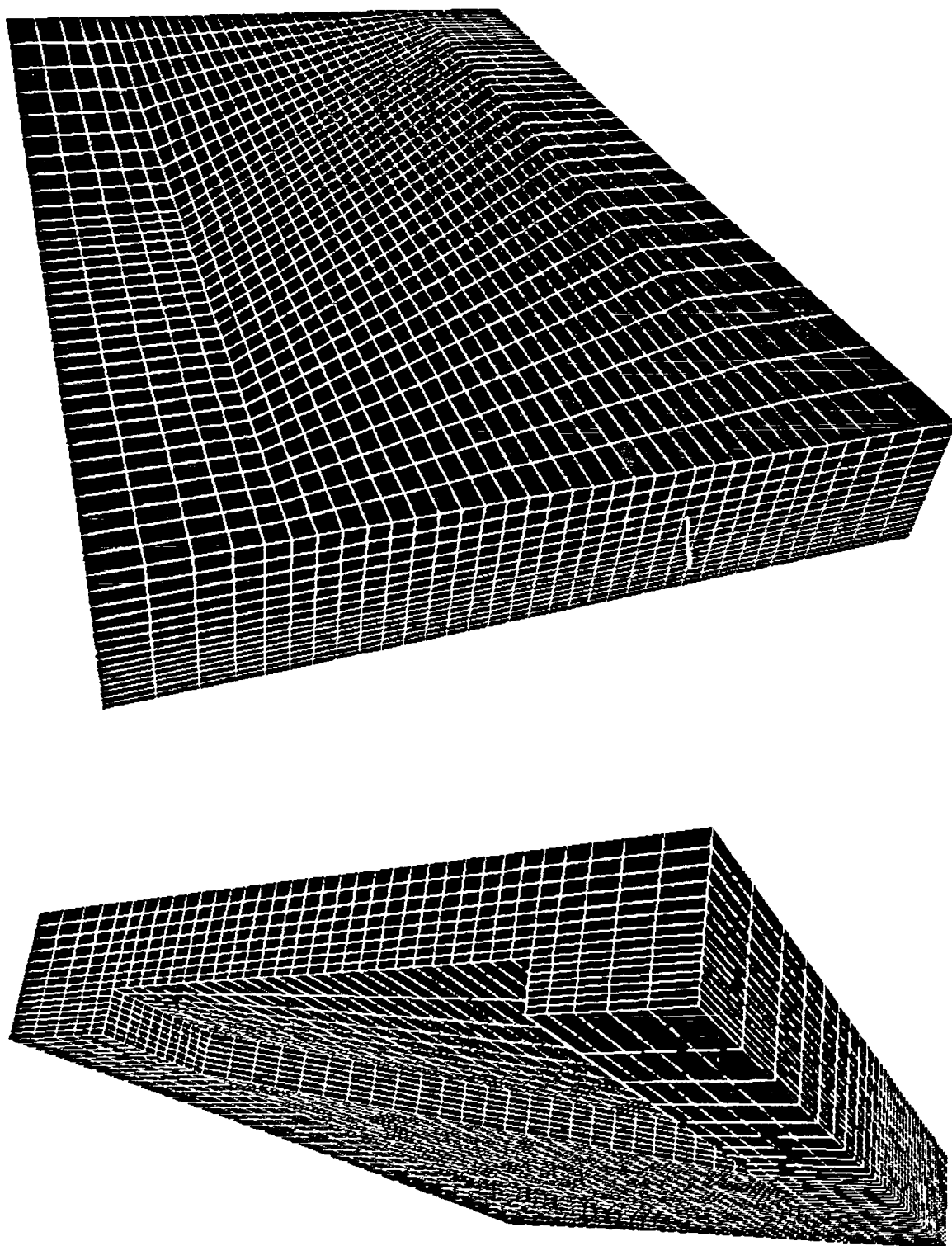


Figure 6.115: 3-D swept backward-facing step: coarse uniform mesh, 16625 CV.

properly resolve such a situation would contain around 1 million cells, which exceeds the computer resources available for this study. In an industrial environment it is also often necessary to resolve the flows which are even more complicated on relatively coarse meshes. This test case can provide an example of the benefits that can be obtained from an adaptive algorithm in such a situation.

The uniform mesh calculations are made on two meshes, with 16625 and 133000 CV-s. The mesh, shown in Fig. 6.115, is slightly graded towards the step and the solid walls.

The stream ribbons in Figs. 6.116 and 6.117 are used to visualise the main features of the flow. In Fig. 6.116 the ribbons are released just above the obstacle and are trapped by the vortex. Their colour represents the local velocity magnitude and their width changes with the magnitude of the vorticity. A single ribbon shown in Fig. 6.117 is released next to the corner of the triangular part and follows the vortex from its creation. The vortex can also be seen in the velocity distribution in the $x - z$ plane on three levels, Figs. 6.118, 6.119 and 6.120. The under-resolved region at the corner of the triangular part in Fig. 6.118 gives an indication of a rapidly changing three-dimensional flow structure over the height of the step. The same conclusion can be reached from the rapid variation of the surface pressure at the same point, shown in Fig. 6.121.

The internal structure of the vortex can be easily seen in the near-surface distribution and the iso-surface of turbulent kinetic energy, Figs. 6.122 and 6.123.

Overall, the gradients in the solution are high for all fields close to the step, with the solution becoming smoother further downstream. It can therefore be expected that the principal error sources will be found along the edge of the step, particularly at the corner of the triangular part.

In the previous test cases error estimation results obtained from the Moment and Residual Error estimates showed good agreement, with the Direct Taylor Series estimate consistently underpredicting the error levels. The exact solution for this test case is again not available and the quality of the estimated error distribution will be judged by comparing the two estimation methods.

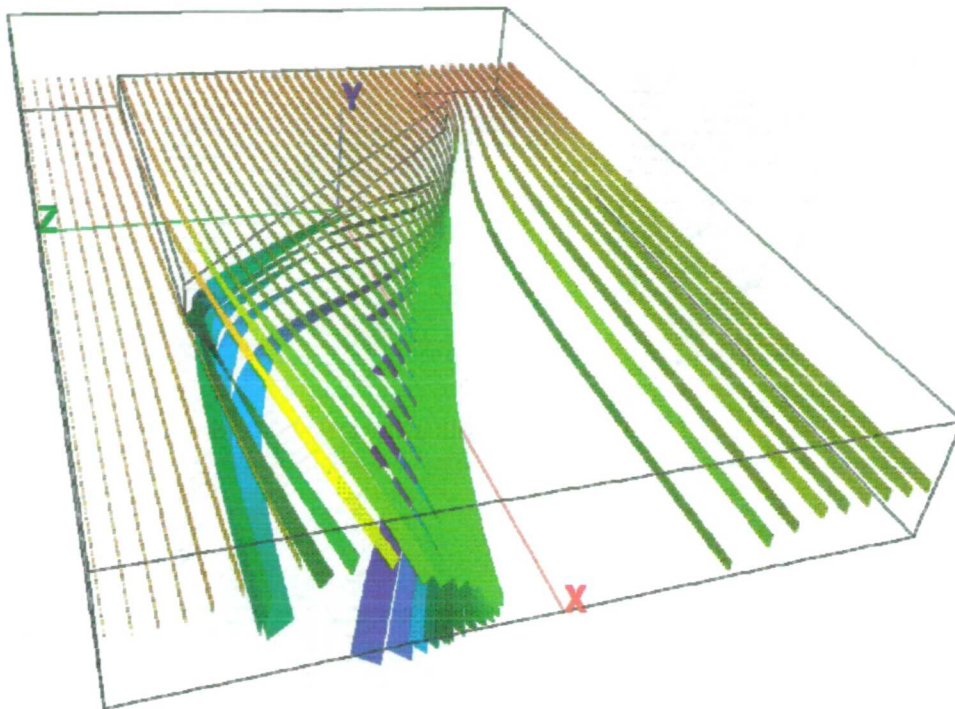


Figure 6.116: 3-D swept backward-facing step: stream ribbons close to the inlet.

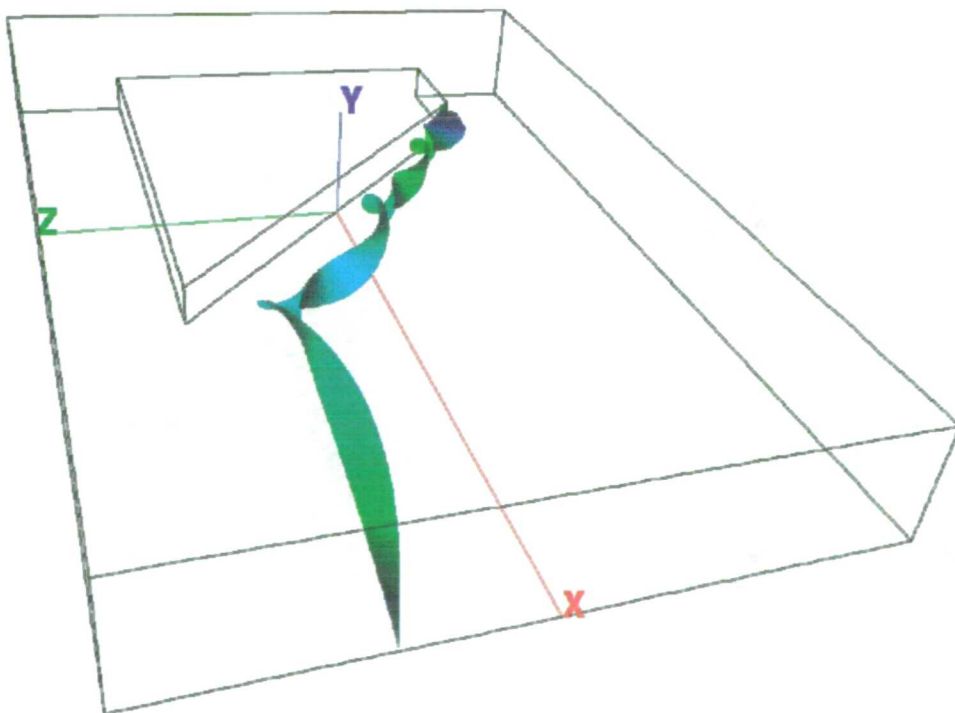


Figure 6.117: 3-D swept backward-facing step: stream ribbon in the vortex.

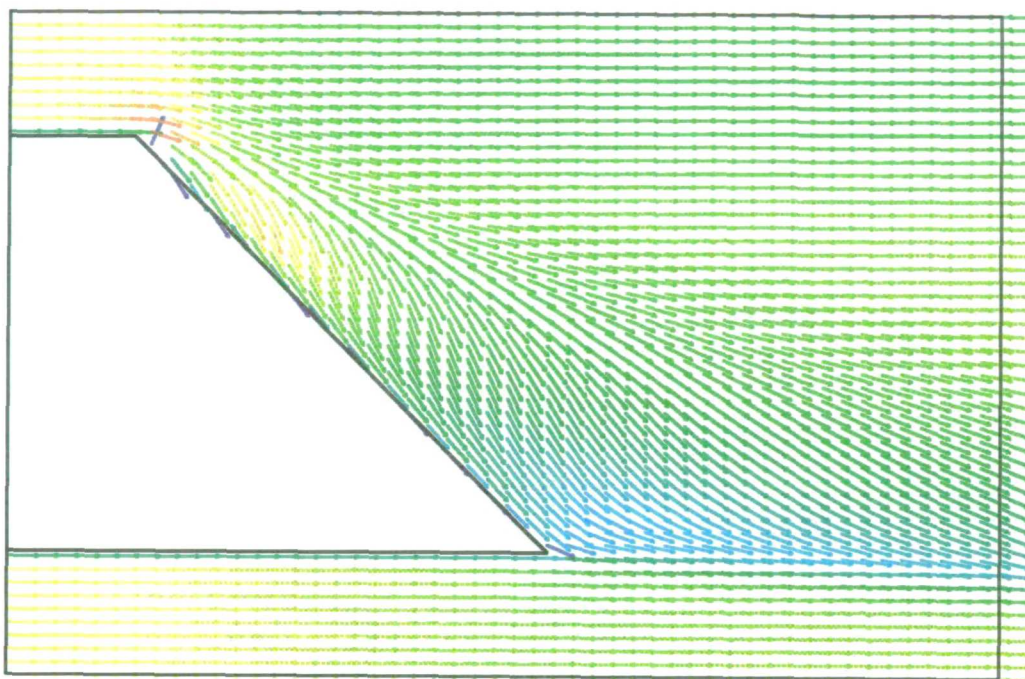


Figure 6.118: 3-D swept backward-facing step: velocity field cut, $y/H = 0.125$.

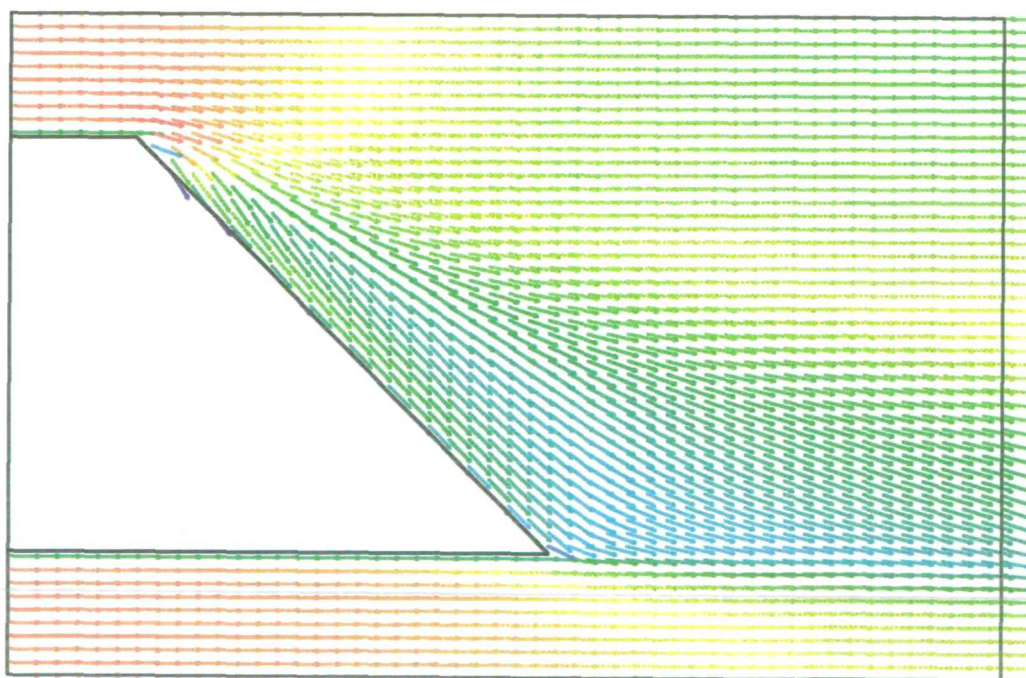


Figure 6.119: 3-D swept backward-facing step: velocity field cut, $y/H = 0.5$.

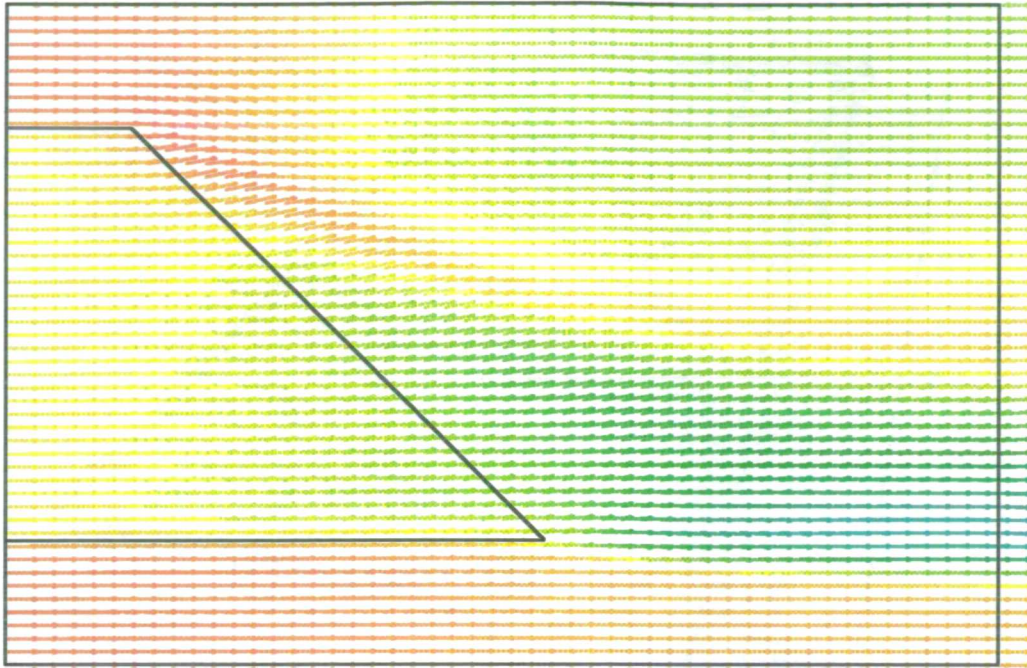


Figure 6.120: 3-D swept backward-facing step: velocity field cut, $y/H = 1.125$.

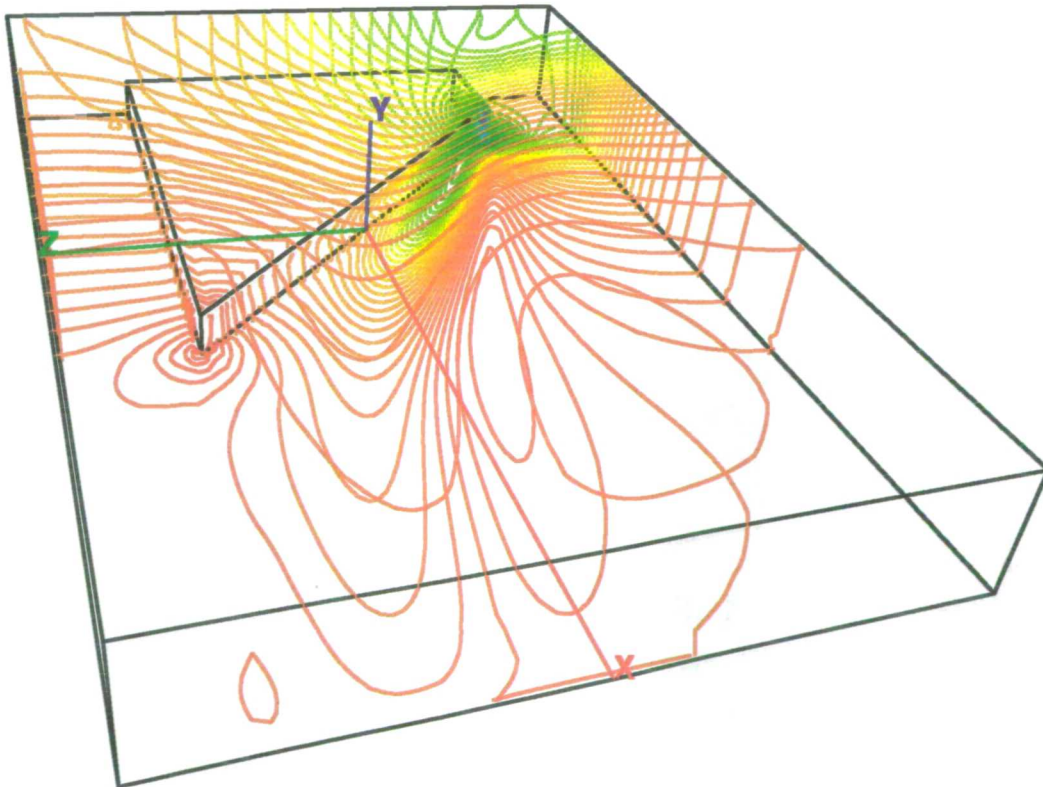


Figure 6.121: 3-D swept backward-facing step: surface pressure.

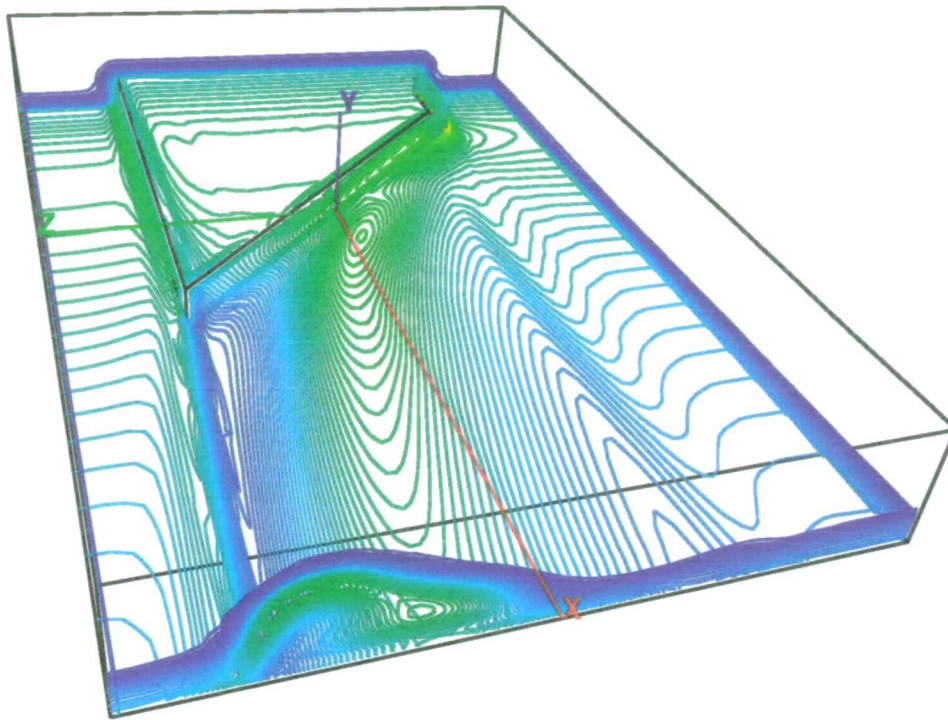


Figure 6.122: 3-D swept backward-facing step: near-surface k distribution.

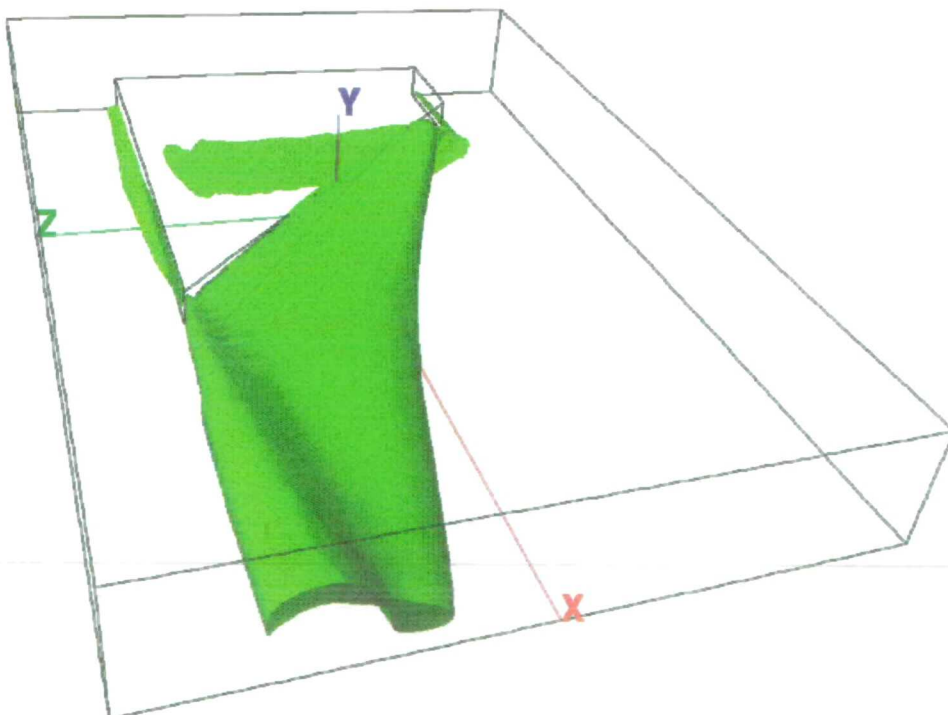


Figure 6.123: 3-D swept backward-facing step: k iso-surface.

The Moment and Residual Error estimates for the velocity field, turbulent kinetic energy and its dissipation are shown in Figs. 6.124, 6.125, 6.126, 6.127, 6.128 and 6.129. The iso-surface level is the same for both error estimates and is given in Table 6.4. The reference value in the table can be used as an indication of the typical magnitude of the variables.

Field	Reference level	Estimated max error	Error iso-surface level
U	$U_0 = 22.5 \text{ m/s}$	4 m/s	0.4 m/s
k	$\bar{k} = 1.5 \text{ m}^2/\text{s}^2$	$1.5 \text{ m}^2/\text{s}^2$	$0.4 \text{ m}^2/\text{s}^2$
ϵ	$200 \text{ m}^2/\text{s}^3$	$150 \text{ m}^2/\text{s}^3$	$50 \text{ m}^2/\text{s}^3$

Table 6.4: 3-D swept backward-facing step: iso-surface level for the Moment and Residual Error estimates.

In the first instance, a close agreement in the error distribution between the two error estimates can be seen for all three fields.

The Moment and Residual Error estimates for velocity highlight several error sources. The highest error is created at the corner of the triangular part, particularly at the bottom of the step. Comparing the error distribution with the velocity cuts (Figs. 6.118, 6.119 and 6.120), the existence of high velocity gradients at the corner of the step is confirmed. Another source of error exists at the downstream edge of the triangle, where the relatively fast side-stream interacts with the vortex. High errors also exist in the zone of high shear behind the top edge of the step. Apart from the above error sources recognised by both error estimates, the Moment Error estimate also highlights several small detached points of high error above the step and downstream from its corner. A look at the mesh structure, Fig. 6.115, reveals that this error originates from the interaction of the mesh skewness and non-orthogonality and high velocity gradients. If the field changes smoothly, the mesh-induced errors are low irrespective of the quality of the mesh.

A similar distribution of the error sources can be seen in the error estimates for the turbulent kinetic energy, shown in Figs. 6.126 and 6.127. Here, the influence of

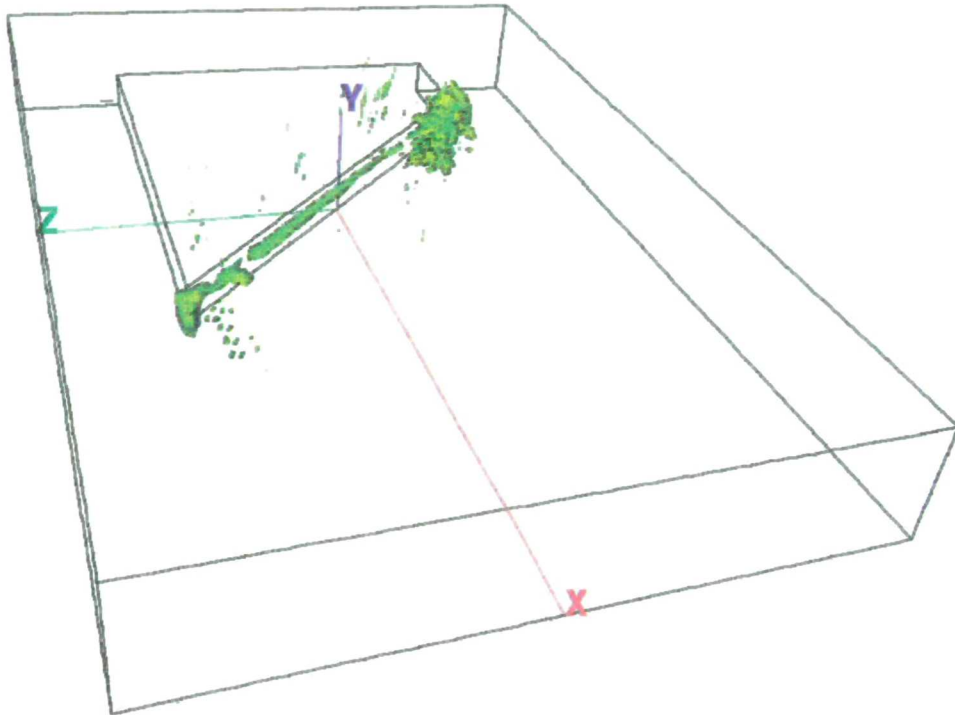


Figure 6.124: 3-D swept backward-facing step: Moment Error estimate for velocity.

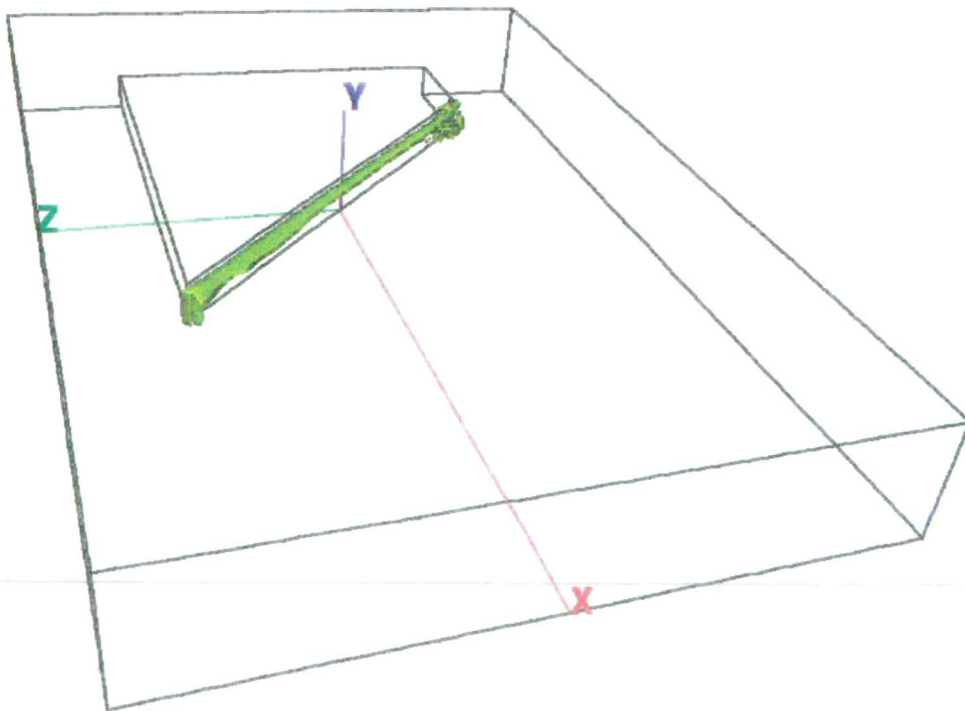


Figure 6.125: 3-D swept backward-facing step: Residual Error estimate for velocity.

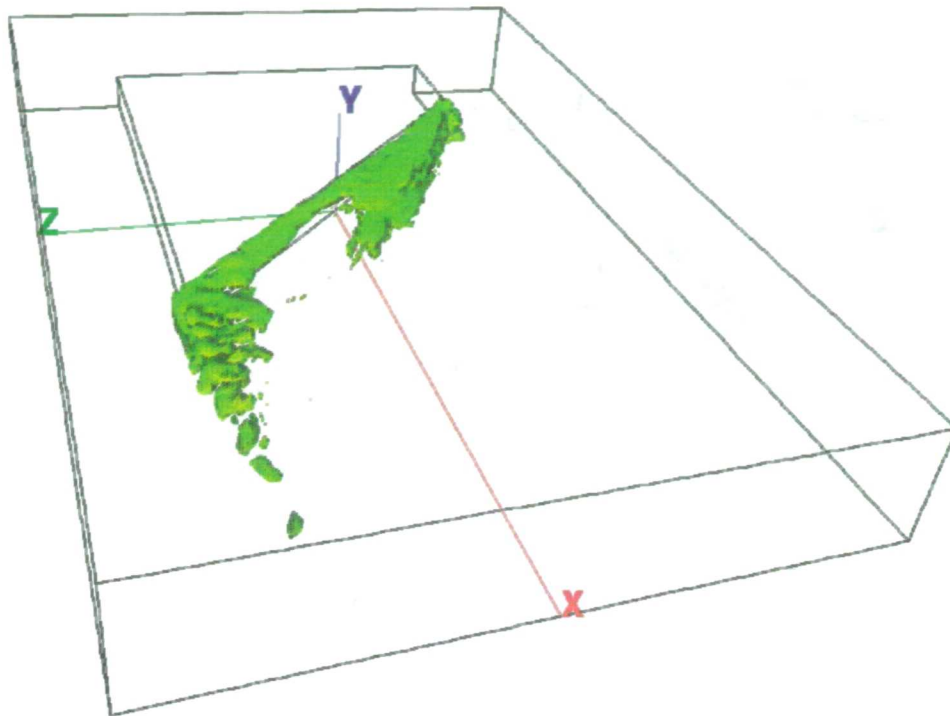


Figure 6.126: 3-D swept backward-facing step: Moment Error estimate for k .

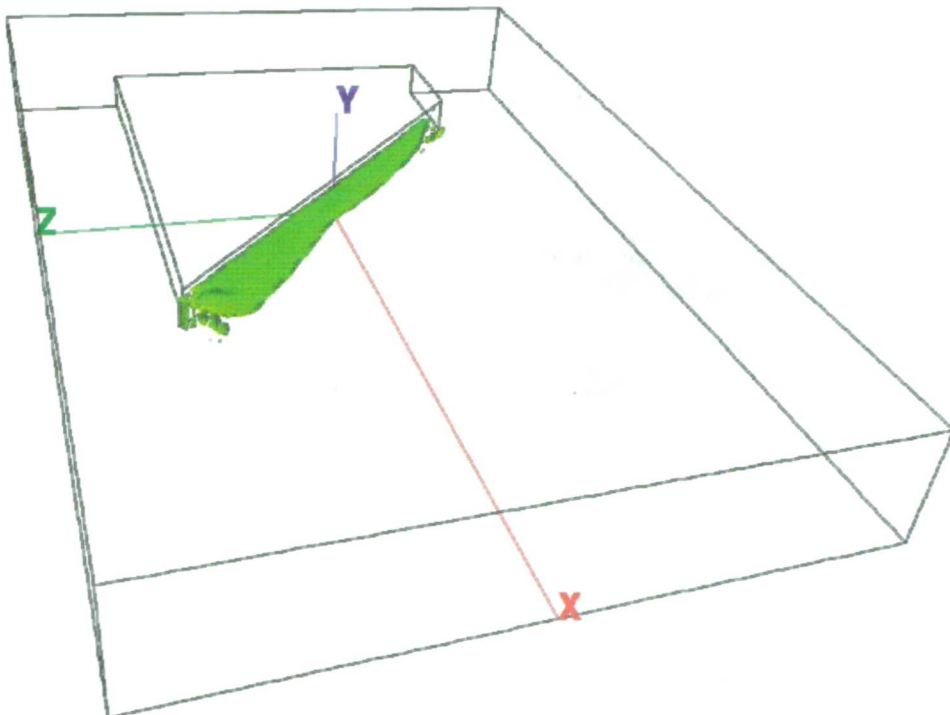


Figure 6.127: 3-D swept backward-facing step: Residual Error estimate for k .

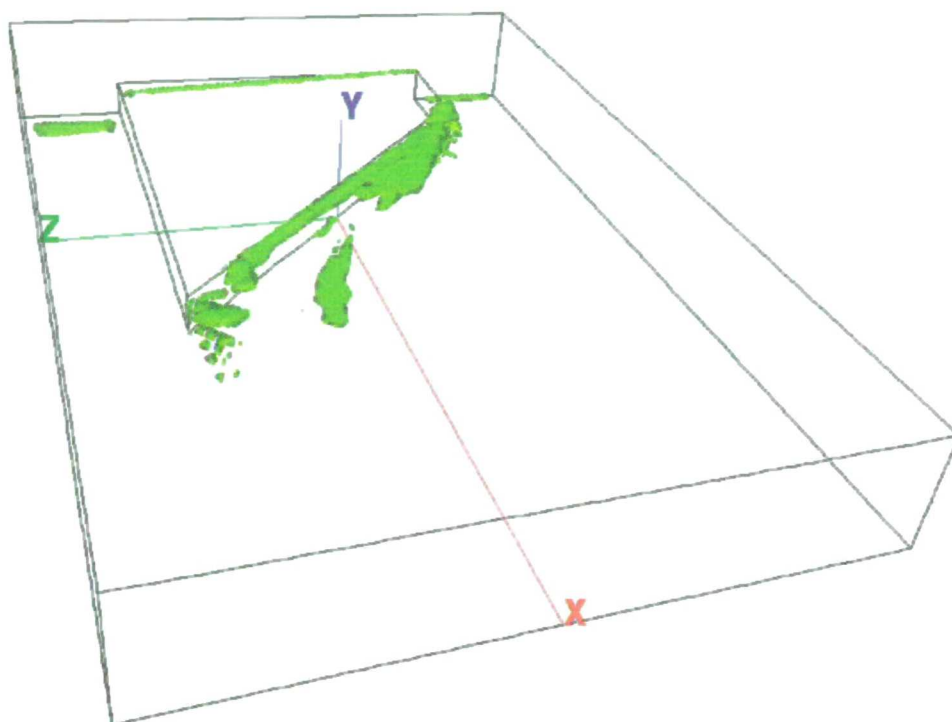


Figure 6.128: 3-D swept backward-facing step: Moment Error estimate for ϵ .

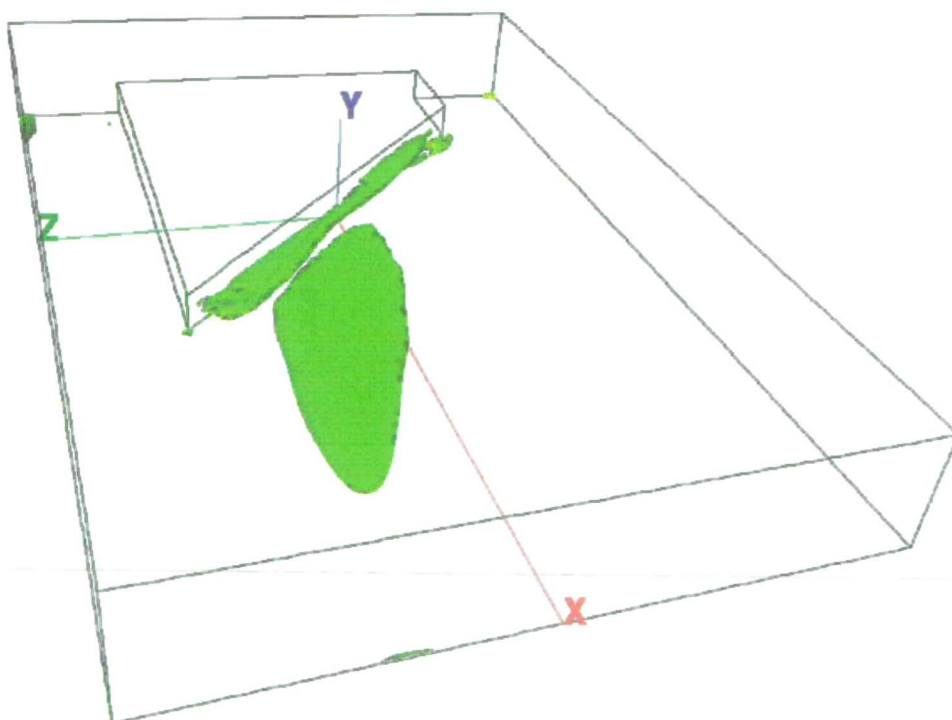


Figure 6.129: 3-D swept backward-facing step: Residual Error estimate for ϵ .

the mesh quality on the accuracy of the solution can be seen even more clearly. The Moment Error estimate shows the “patchy” error distribution spreading behind the edge of the triangular part, decreasing in size as the k distribution becomes smoother. The error is again high along the top of the step and at the corners of the triangle. The Moment Error estimate also follows the early part of the vortex, where the generation of k is high. The Residual Error estimate predicts the lower magnitude of the error at the upstream corner relative to the Moment Error estimate, creating a slight discrepancy between the estimates.

Error estimates for ϵ are shown in Figs. 6.128 and 6.129. Apart from the high error levels along the edge of the step, both the Moment and Residual Error estimate detect an additional error source further downstream. The origin of this can be explained using similar calculations in two spatial dimensions. The main feature of the ϵ distribution in the backward-facing step geometries is a long and narrow region of high dissipation just behind the edge of the step, characterised by high gradients in ϵ . The width of this feature is of the order of $0.2 H$, which means that the mesh with 133000 CV-s resolves it over approximately three cells. This is clearly not good enough to resolve the high gradients in ϵ and causes high local errors.

Two refinement strategies will now be used: refinement-only from the initial mesh with 16625 CV-s (Fig. 6.115) and refinement/unrefinement from the mesh with 133000 CV-s. According to the error levels shown above, the uniform mesh used for refinement/unrefinement is still not very fine. The potential benefit from unrefinement will therefore be limited. A single level of mesh adaptation in this calculation creates a mesh with 301496 CV-s, reaching the limit of the available computer resources.

In order to demonstrate at least three levels of refinement without hitting the limit of the available computer resources, the refinement-only calculation uses the following refinement parameters:

$$\lambda_{ref} = 8$$

$$\xi = 1.2$$

The refinement/unrefinement uses the values recommended in Chapter 5:

$$\lambda_{ref} = 1.5$$

$$\lambda_{ref} = 0.5$$

$$\xi = 1.2$$

In order to preserve the accuracy of wall-functions, the y^+ in the near-wall cell is not allowed to drop below 50. The resolution of the initial mesh is so poor that only one cell has been blocked from refinement for this reason during both adaptive calculations.

The refinement patterns in 3-D extend behind the step all the way along its width. In order to provide an insight into the mesh structure, a portion of the domain close to the upstream edge of the triangle is selected. Error estimates predict high errors around this corner, which will result in extensive mesh changes.

The uniform mesh around the corner of the step is shown in Fig. 6.130. The height of the step is split into only seven control volumes, with the y^+ value for the near-wall cell of the order of 500 or more. This mesh is only able to pick up the basic features of the flow, without any hope of producing accurate results. The first level of refinement, Fig. 6.131, doubles the number of cells over the height of the step and provides considerably better resolution around the corner. The last two cells at the top of the step have also been refined. The two consequent refinement levels, Figs. 6.132 and 6.133, follow the same trend. After three levels of refinement, the step height is divided into 46 CV-s, a considerable improvement in comparison with the initial uniform mesh.

Table 6.5 gives the change in the number of cells through the levels of adaptive refinement, giving the number of cells in the refined mesh as a percentage of the size of the uniform mesh giving equivalent local resolution in regions of high error.

In order to demonstrate the improvement in accuracy through the refinement procedure, we shall follow the changes in the error iso-surface on the four meshes. Figs. 6.134, 6.135, 6.136 and 6.137 shows the iso-surface of $0.75 m/s$ in the Momentum Error estimate for velocity. Initially, the high error envelopes both corners of the

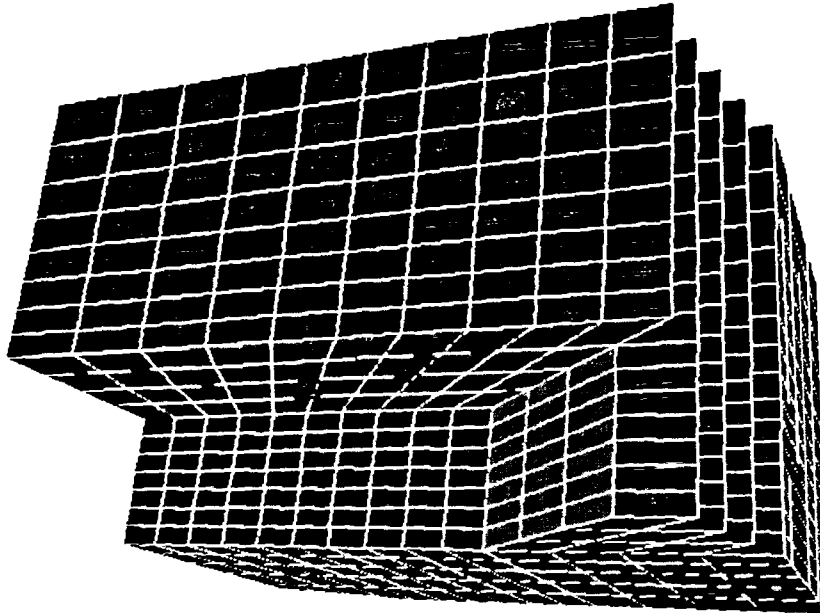


Figure 6.130: 3-D swept backward-facing step: coarse uniform mesh at the corner of the triangular part.

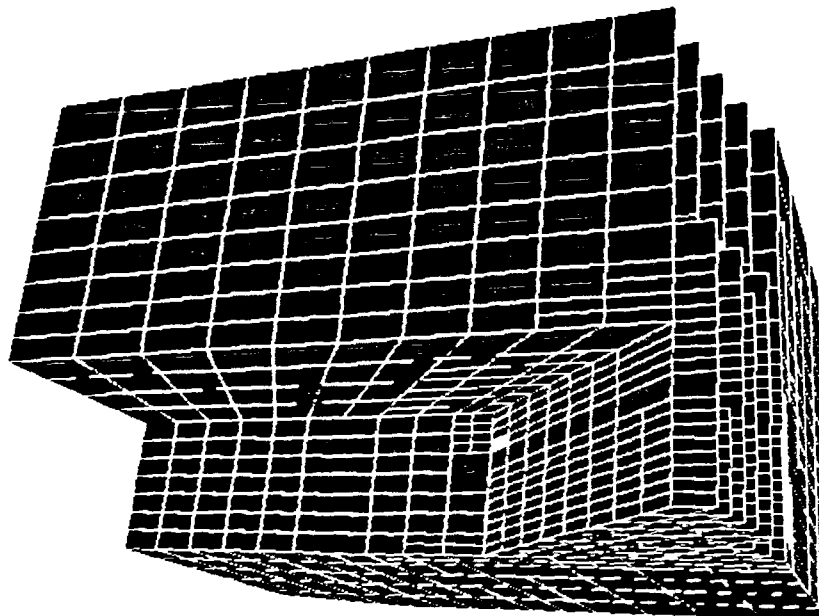


Figure 6.131: 3-D swept backward-facing step: mesh detail after the first level of refinement.

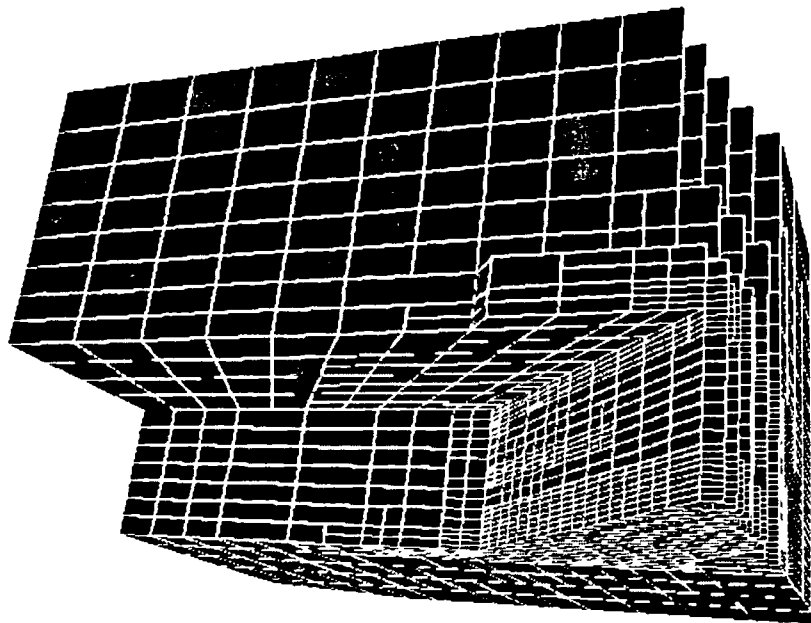


Figure 6.132: 3-D swept backward-facing step: mesh detail after the second level of refinement.

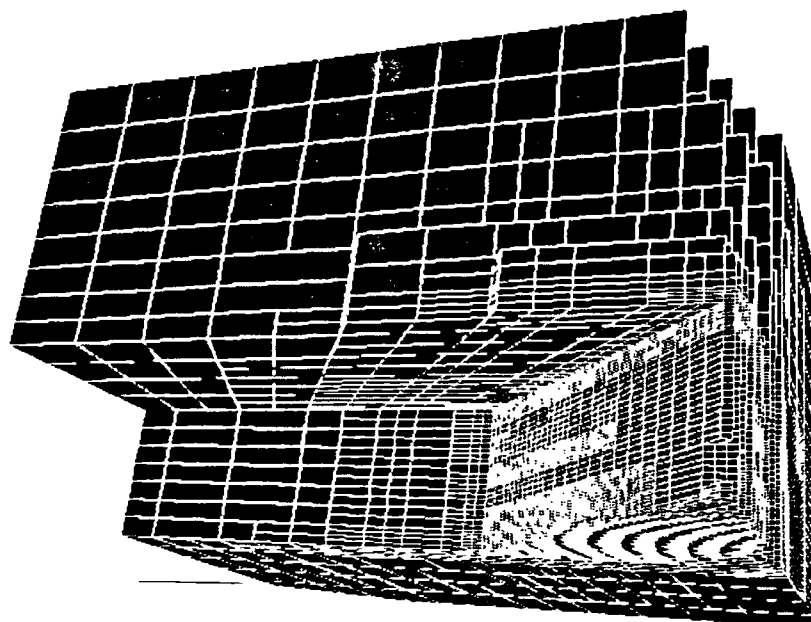


Figure 6.133: 3-D swept backward-facing step: mesh detail after the third level of refinement.

Adaptive refinement	% ratio	Uniform
16625	100 %	16625
27563	20.7 %	133000
64003	6.01 %	1064000
202957	2.38 %	8512000

Table 6.5: 3-D swept backward-facing step: number of cells for adaptive and uniform refinement starting from the coarse mesh.

triangle, also showing several points of high mesh-induced errors behind the step. As the refinement progresses, the region of high error becomes smaller and closer to the solid walls. The mesh-induced error is moved to the point where the refined part interacts with the coarse mesh away from the step. In the final mesh, the Moment Error estimate is high just at the tip of the triangle and close to the wall at the upstream edge.

The changes in the Residual Error estimate for k are shown in Figs. 6.138, 6.139, 6.140 and 6.141, with the iso-surface of $0.35 \text{ m}^2/\text{s}$. The error is initially high inside the vortex, downstream from the step. As the refinement progresses, the error peak moves closer to the wall and the top edge.

In terms of the maximum error reduction, neither uniform nor adaptive refinement show the expected scaling. The change in the maximum error in velocity for the adaptive calculation is shown in Fig. 6.142. All error estimates predict an increase in its magnitude as the mesh becomes finer. The increase is highest in the Direct Taylor Series Error estimate: as the mesh refinement picks up the new features of the solution the second velocity gradient increases, pushing the error estimate higher. The maximum error for the Residual Error estimate is given both with and without the near-wall errors. The near-wall error is higher than for the Moment Error estimate, as was the case for the turbulent flow over a hill. When the near-wall cells are removed from the comparison, the Moment and Residual Error estimates show a reasonable agreement. It is interesting to notice that in spite of

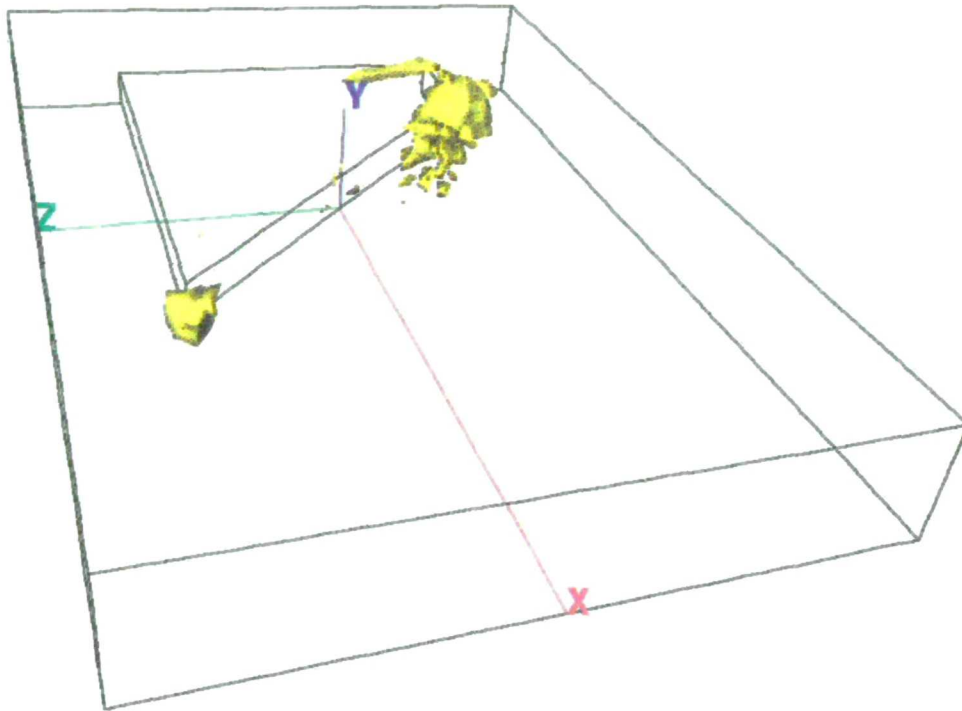


Figure 6.134: 3-D swept backward-facing step: iso-surface of the Moment Error estimate for velocity on the initial mesh.

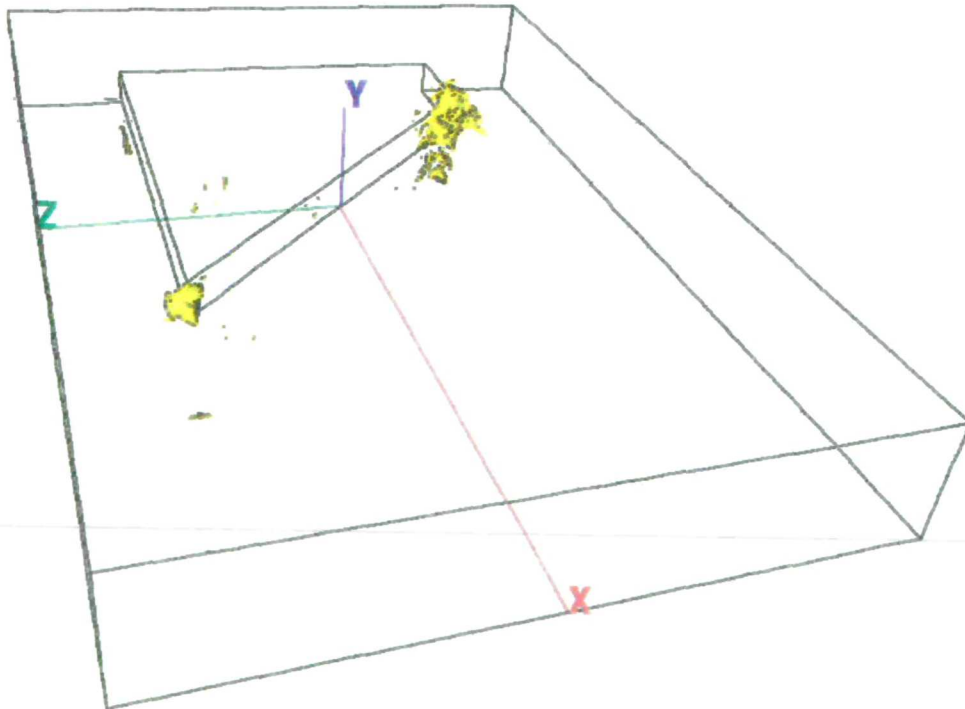


Figure 6.135: 3-D swept backward-facing step: iso-surface of the Moment Error estimate for velocity after the first level of refinement.

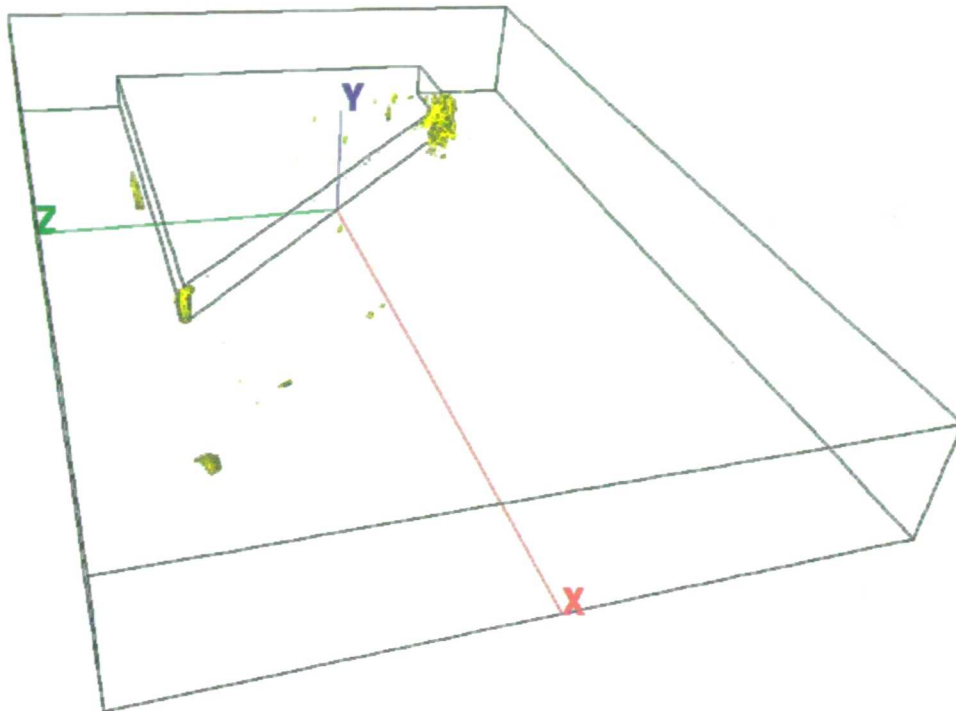


Figure 6.136: 3-D swept backward-facing step: iso-surface of the Moment Error estimate for velocity after the second level of refinement.

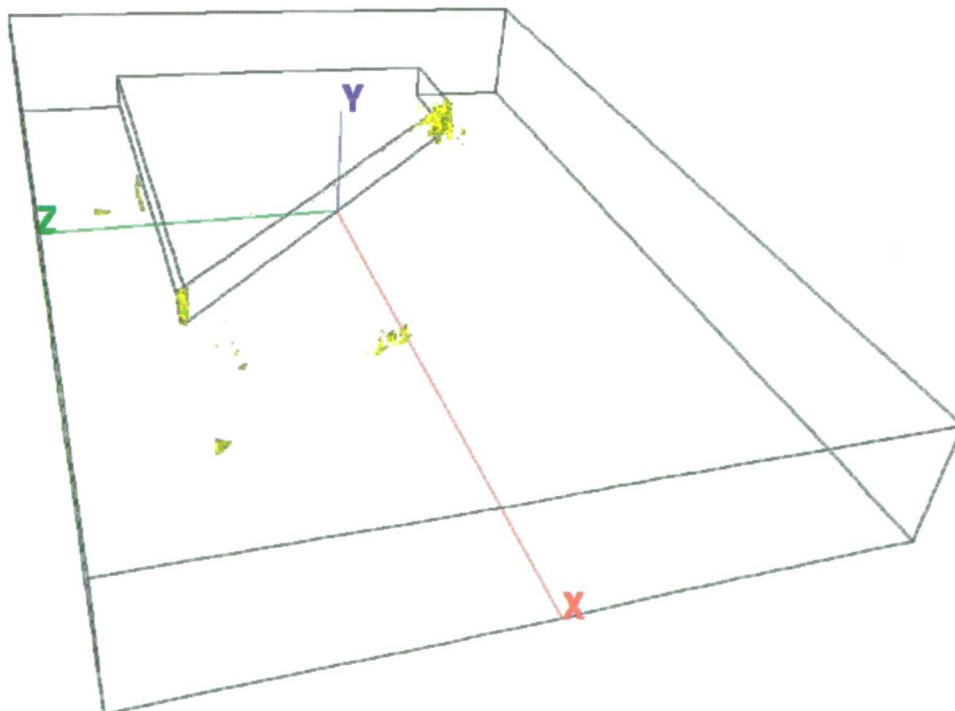


Figure 6.137: 3-D swept backward-facing step: iso-surface of the Moment Error estimate for velocity after the third level of refinement.

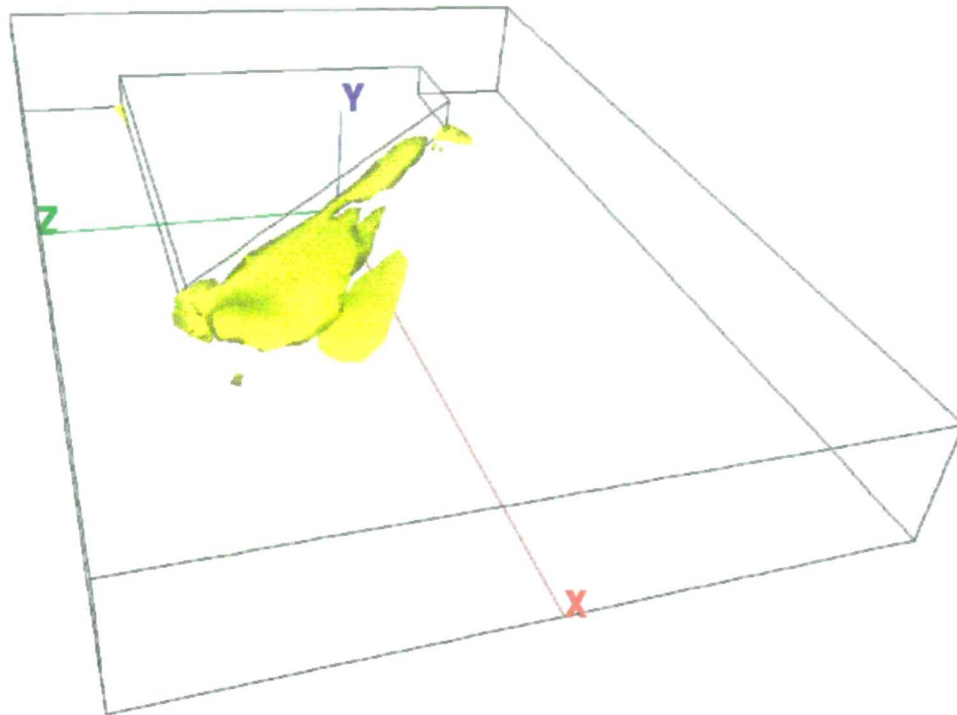


Figure 6.138: 3-D swept backward-facing step: iso-surface of the Residual Error estimate for k on the initial mesh.

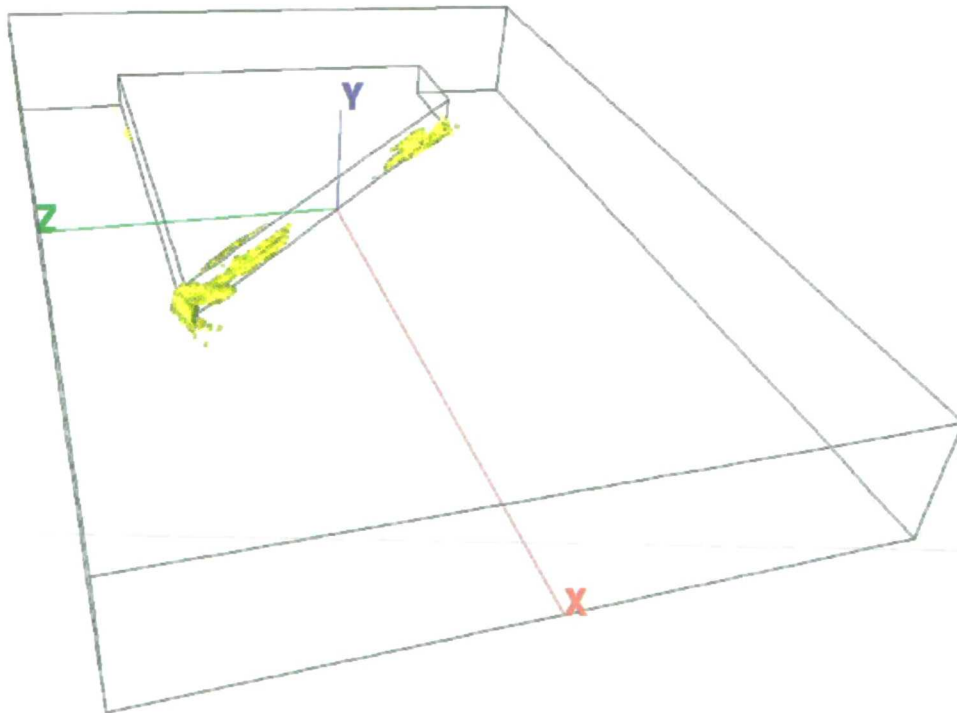


Figure 6.139: 3-D swept backward-facing step: iso-surface of the Residual Error estimate for k after the first level of refinement.

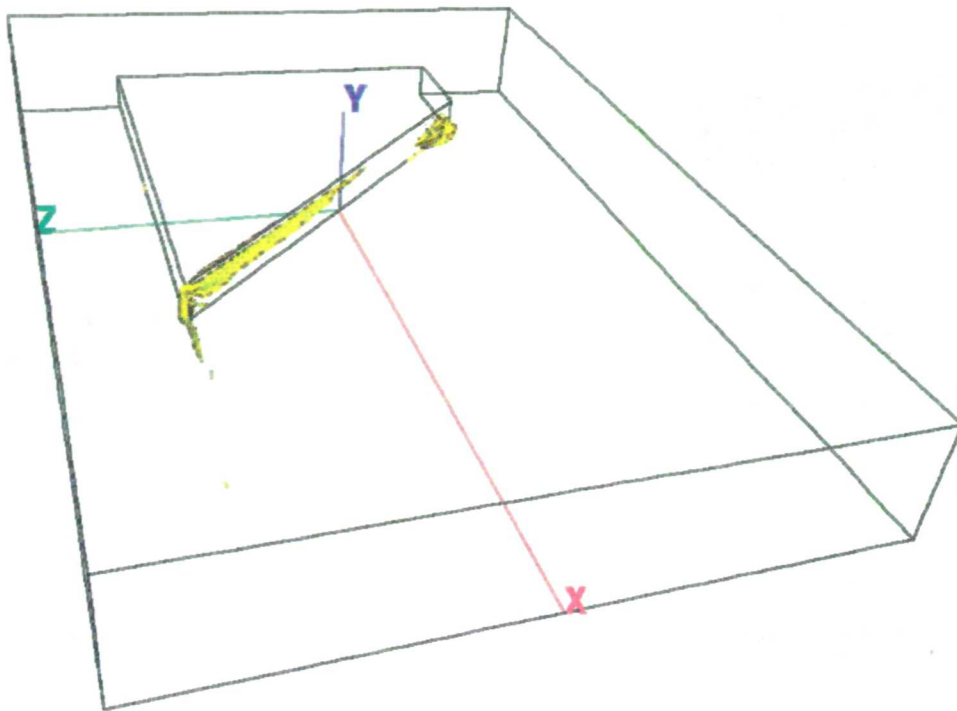


Figure 6.140: 3-D swept backward-facing step: iso-surface of the Residual Error estimate for k after the second level of refinement.

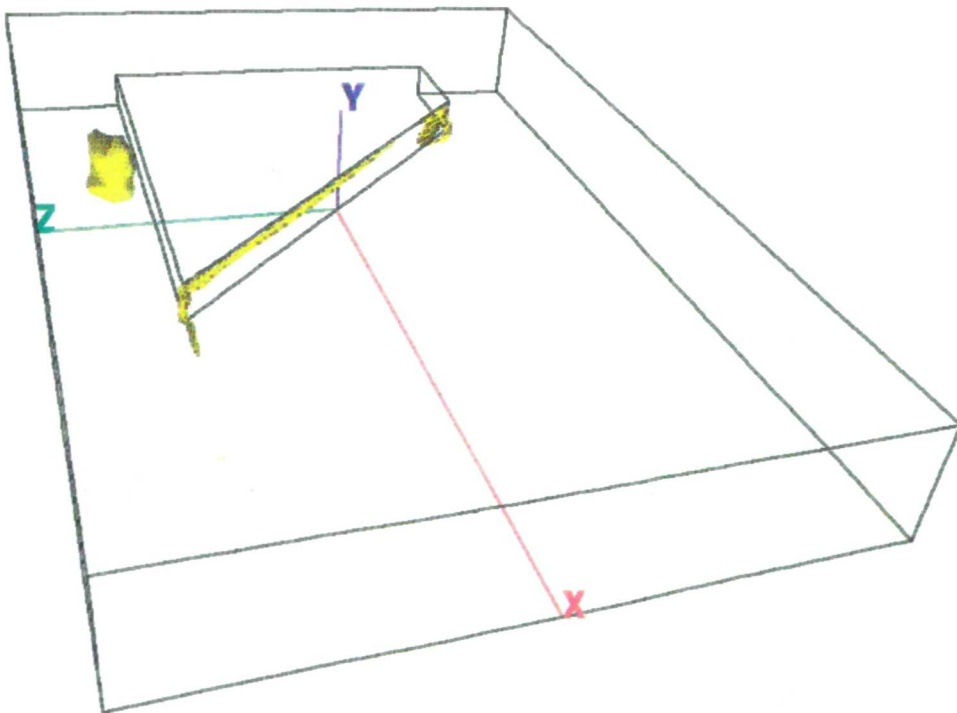


Figure 6.141: 3-D swept backward-facing step: iso-surface of the Residual Error estimate for k after the third level of refinement.

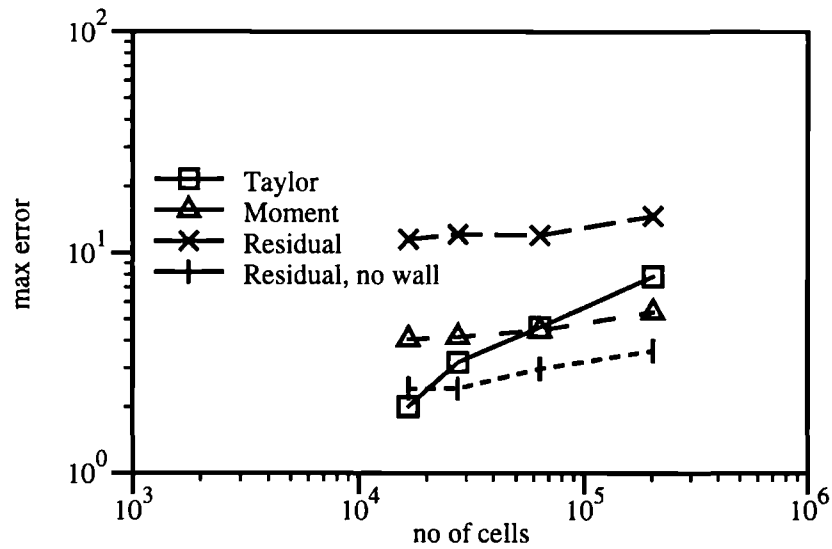


Figure 6.142: 3-D swept backward-facing step: scaling of the maximum velocity error for adaptive refinement.

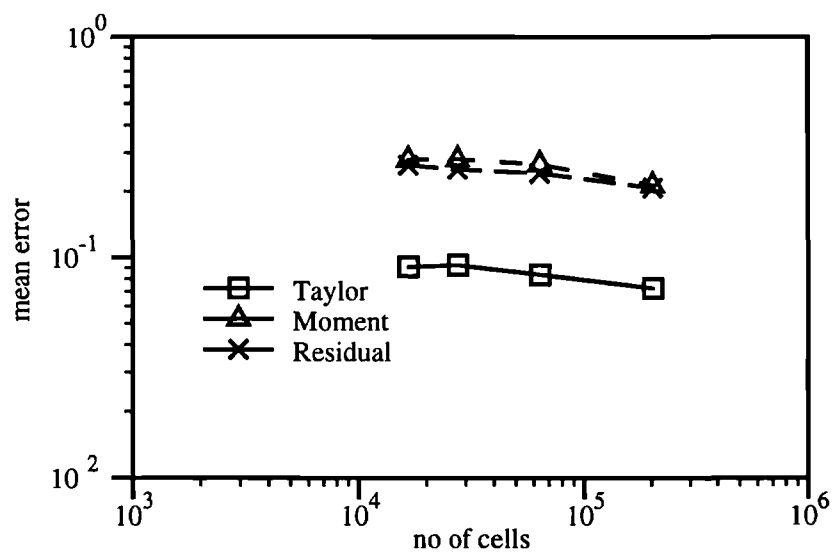


Figure 6.143: 3 D sw pt backward facing step: scaling of the mean velocity error for adaptive refinement.

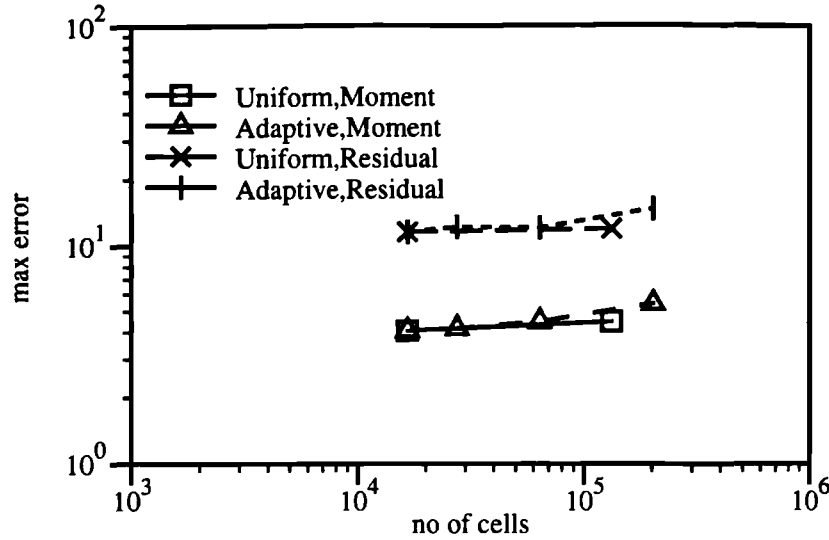


Figure 6.144: 3-D swept backward-facing step: scaling of the maximum velocity error for uniform and adaptive refinement.

the sampling bias and the increase in the maximum error, the mean velocity error (Fig. 6.143) drops according to all error estimates.

The difference in the error reduction for velocity between the uniform and adaptive refinement is given in Fig. 6.144 for the Moment and Residual Error estimates. The first level of uniform and adaptive refinement create the same maximum errors. As the adaptive procedure progresses, the maximum velocity error increases, giving an indication of the insufficient resolution of both uniform meshes.

The change in the maximum error for k , shown in Fig. 6.146 is typical for adaptive calculations starting from very coarse meshes. Early stages of refinement reveal previously invisible flow features, causing an increase in the error level. When the mesh resolution becomes sufficient to pick up all the details of the flow, the maximum error starts to drop. The adaptive refinement gives a qualitative advantage over the initial (uniform mesh) solution in spite of the fact that the maximum error is higher. The cause behind the low initial error is a completely inappropriate mesh resolution, which creates a wrong picture of the complexity of the flow field. The adaptively refined mesh, on the other hand, resolves all the features of the flow. A comparison of the error for uniform and adaptive refinement, Fig. 6.145, reveals that

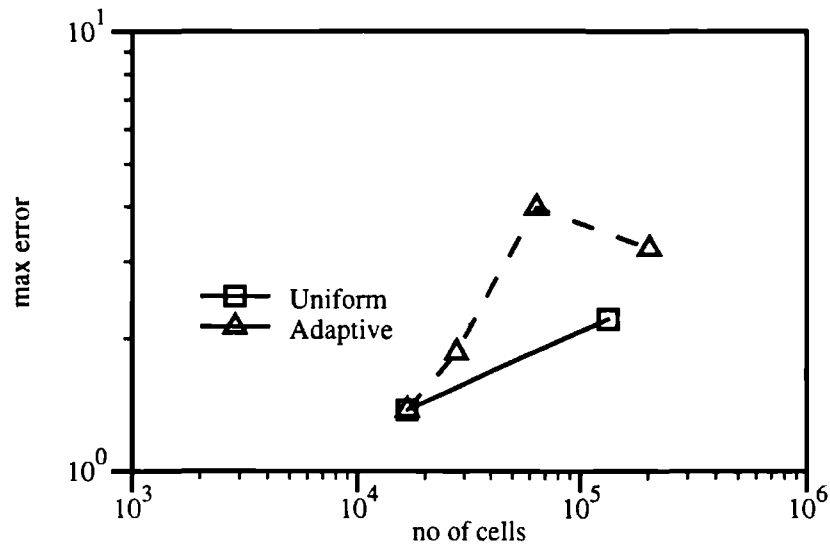


Figure 6.145: 3-D swept backward-facing step: scaling of the maximum k error for uniform and adaptive refinement.

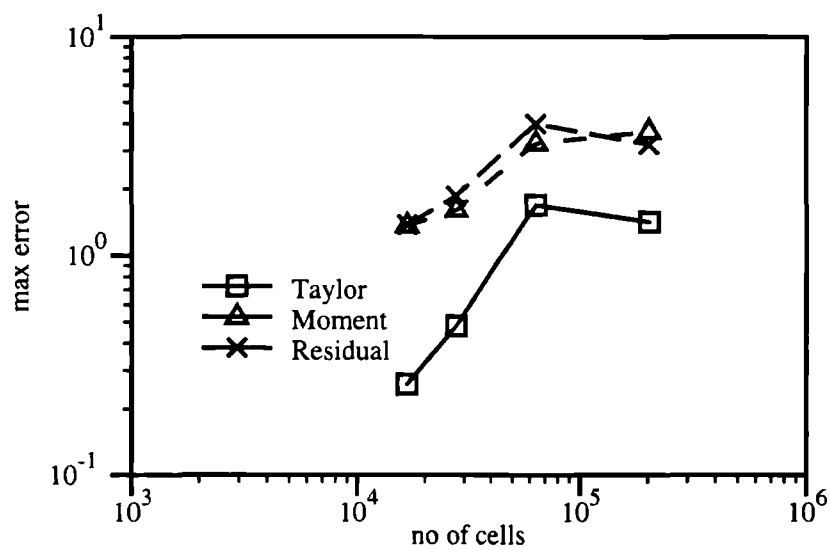


Figure 6.146: 3-D swept backward-facing step. scaling of the maximum k error for adaptive refinement.

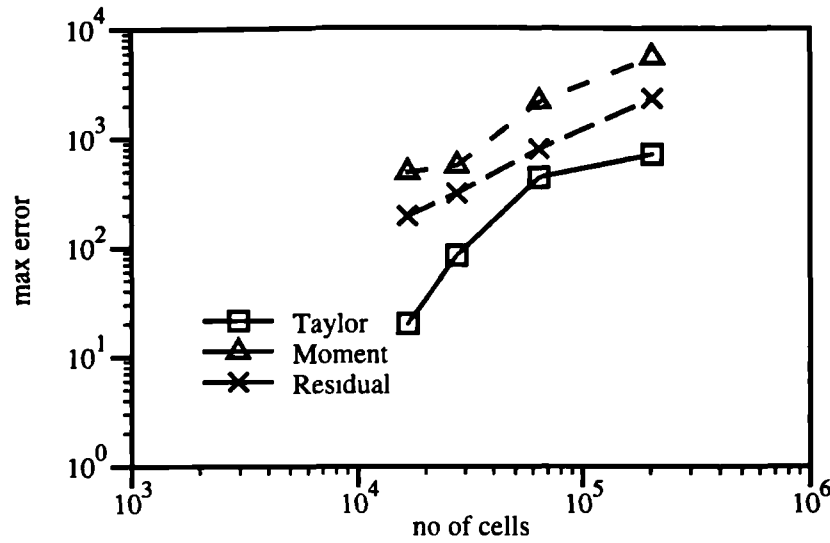


Figure 6.147: 3-D swept backward-facing step: scaling of the maximum ϵ error for adaptive refinement.

the uniform procedure needs another two levels of refinement to start reducing the maximum error in k . Such a mesh would consist of 8512000 CV-s, 42 times more than the corresponding adaptively refined mesh.

The scaling of the maximum error for the proposed error estimates in the adaptive procedure is given in Fig. 6.147. Mesh adaptivity decreases the y^+ value in the near-wall cell, resulting in an increase in the near-wall ϵ levels. In spite of the fact that the estimated error in this cell is set to zero, all error estimates show a rapid increase in the maximum error. The error in this case comes from the fact that a high ϵ near the wall induces high gradients towards the bulk of the flow, requiring more mesh resolution. This error cannot be overcome in an appropriate way, as ϵ and its near-wall gradients rise faster than the newly added mesh resolution can resolve them.

The refinement-only and refinement/unrefinement calculations reach the limit of the available computer resources after only a single level of mesh adaptation. As the mesh with 133000 CV-s is still not appropriate in terms of resolution, the errors behave in much the same way as in refinement-only from the coarse mesh. The mesh size for both refinement-only and refinement/unrefinement are given in Table 6.6.

Refinement only	% ratio	Refinement and unrefinement	% ratio	Uniform
133000	100 %	133000	100 %	133000
323958	30.4 %	301496	28.3 %	1064000

Table 6.6: 3-D swept backward-facing step: number of cells for refinement-only and refinement/unrefinement starting from the fine mesh.

The benefit from unrefinement in this case is quite limited, because of the low initial resolution and only a single refinement level.

In summary, this test case shows much the same behaviour as the other turbulent test case presented in this Chapter. The error reduction is slow for both uniform and adaptive mesh refinement, with most of the problems caused by the treatment of the wall. The mesh size necessary to appropriately resolve three-dimensional flows is much larger than in 2-D. It can therefore be expected that in industrial applications the user needs to rely on comparatively coarse meshes. In such situations, the benefit of an adaptive procedure should be described more in terms of appropriate resolution of the details of the solution than in the reduction of the error.

Considering the size of three-dimensional problems, the issue of efficiency of the adaptive algorithm on vector and parallel computer architectures becomes important. The error estimation methods presented in this study operate on a cell-by-cell basis and therefore vectorise and parallelise naturally. The mesh refinement procedure needs to be split into two parts. The first, selection of the type of the cell split does not vectorise (it is based on non-vectorisable topological analysis), but can be parallelised with the use of inter-processor face communication. The second part includes the actual creation of the new mesh and can again be parallelised with a certain amount of effort. It would, however, be very difficult to automatically keep the load balance within reasonable limits. The refinement in this study has therefore been performed in a single-processor mode, followed by the mesh decomposition which produces perfect load balancing irrespective of the mesh structure.

6.5 Vortex Shedding Behind a Cylinder

The final test case in this Chapter provides an insight into the properties of error estimation and adaptivity in transient calculations. For this purpose, the laminar shedding flow behind a cylinder in two spatial dimensions will be used. The test setup consists of a cylinder with the radius $r = 0.01\text{ m}$ in a channel, Fig. 6.148. The

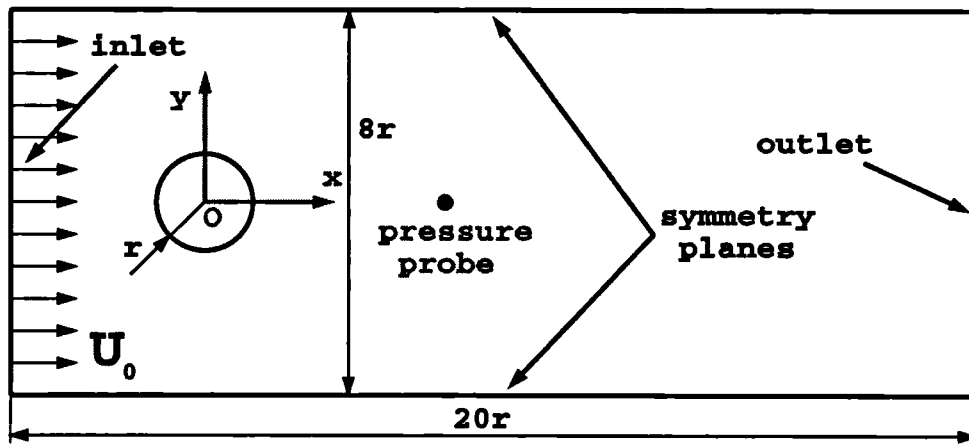


Figure 6.148: Vortex shedding behind a cylinder: test setup.

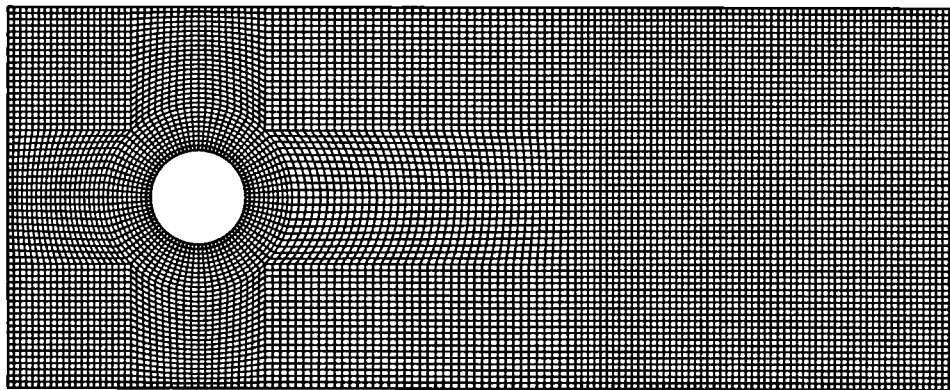


Figure 6.149: Vortex shedding: uniform mesh.

inlet velocity is set to $U_0 = 1\text{ m/s}$ and with the kinematic viscosity of $5 \times 10^{-5}\text{ m}^2/\text{s}$, gives a Reynolds number of 400. On the top and bottom boundaries of the channel, a symmetry plane boundary condition is used. The initial mesh consists of 8800 CV-s and is shown in Fig. 6.149.

This flow is characterised by the shedding of vortices behind the cylinder, which can be seen in velocity and pressure fields in Figs. 6.150 and 6.151. The vortical

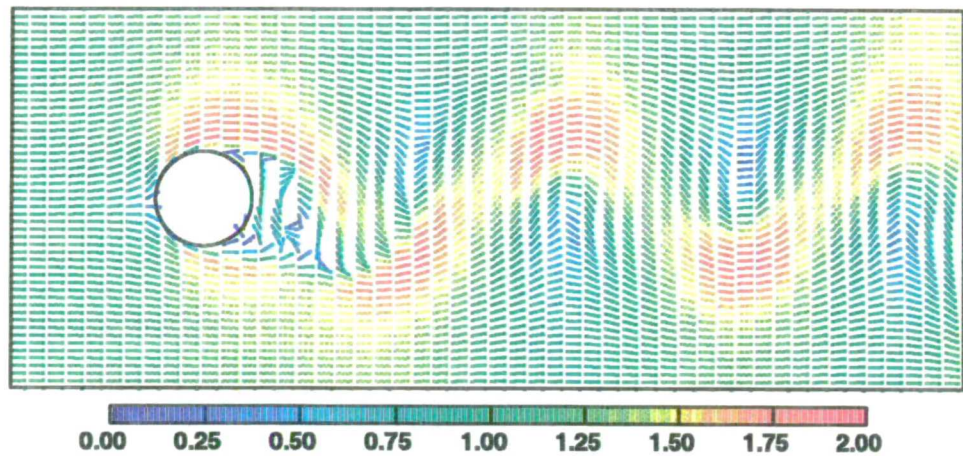


Figure 6.150: Vortex shedding: velocity field.

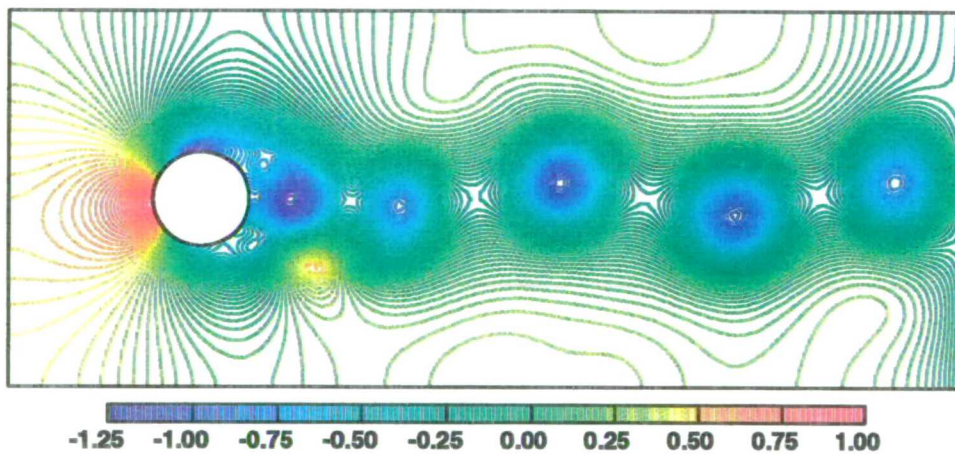


Figure 6.151: Vortex shedding: pressure field.

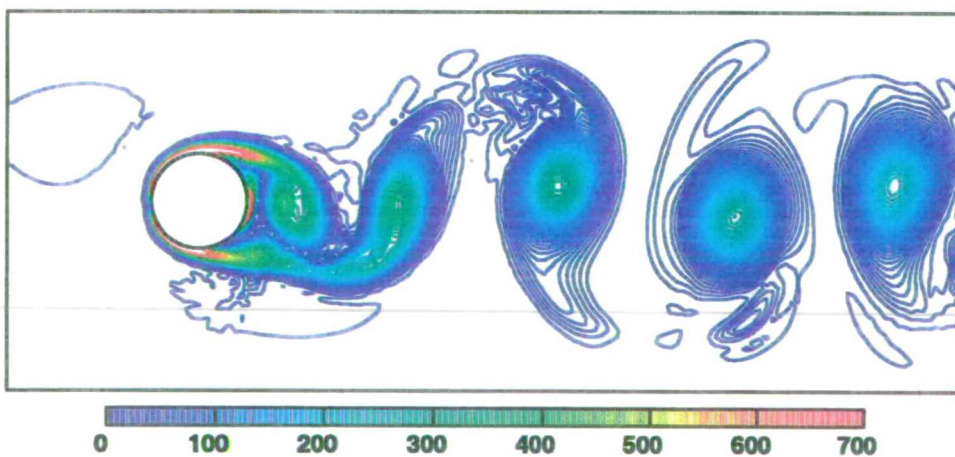


Figure 6.152: Vortex shedding: enstrophy distribution.

motion is also visible in the distribution of the enstrophy (magnitude of the curl of the velocity), shown in Fig. 6.152.

In the first instance, the flow will be resolved on the same mesh with three temporal discretisation schemes, namely the first-order Euler Implicit (EI) discretisation and two second-order schemes: Backward Differencing in time (BD) and the Crank-Nicholson (CN) method. The temporal changes will be followed in the variation of the pressure in the point 0.05 m downstream from the origin (see Fig. 6.148). Initially, the time-step is set to $2 \times 10^{-4}\text{ s}$, giving a maximum Courant number of 0.4. The pressure traces for the three methods of temporal discretisation are given in Fig. 6.153. The results for the two second-order accurate temporal discretisation

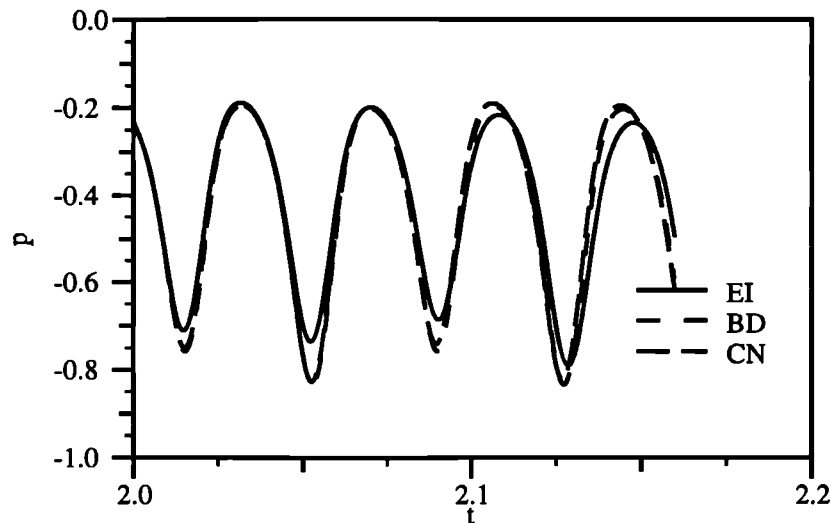


Figure 6.153: Vortex shedding: pressure trace for different methods of temporal discretisation.

methods are very close to each other, with the Euler Implicit discretisation showing small discrepancies in both the pressure amplitudes and the length of the period. The accumulation of the temporal error can be seen even on this slowly evolving flow³.

The Residual Error estimate for transient flows described in Section 4.7 will now be used. The distribution of the full Residual Error estimate for this case is shown

³In this test case, a single pressure cycle is resolved over 200 time-steps.

in Fig. 6.154. The principle error source is located on the surface of the cylinder and is caused by the high velocity gradients in the boundary layer. The error on the windward side remains more or less constant in time. The other error peak is associated with the point where the vortex is created. Here, the high error oscillates between the top and bottom half of the cylinder, depending on the position of the vortex.

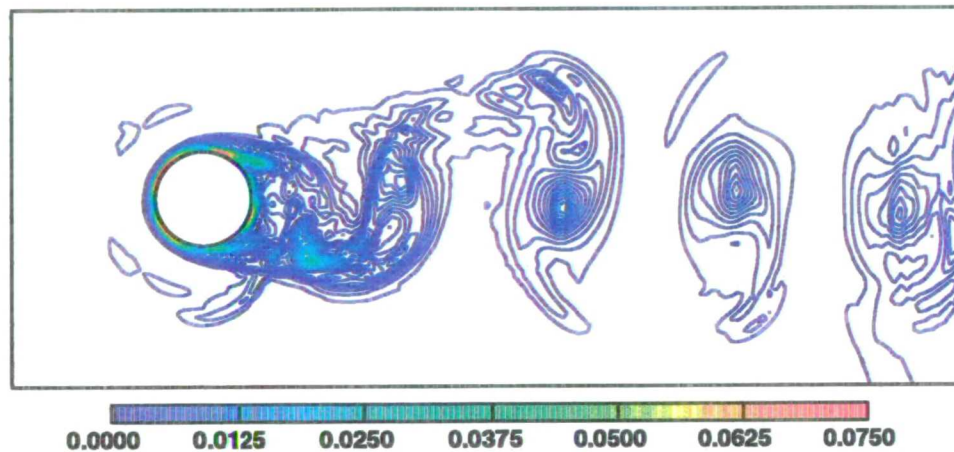


Figure 6.154: Vortex shedding: full Residual Error for EI on $Co = 0.4$.

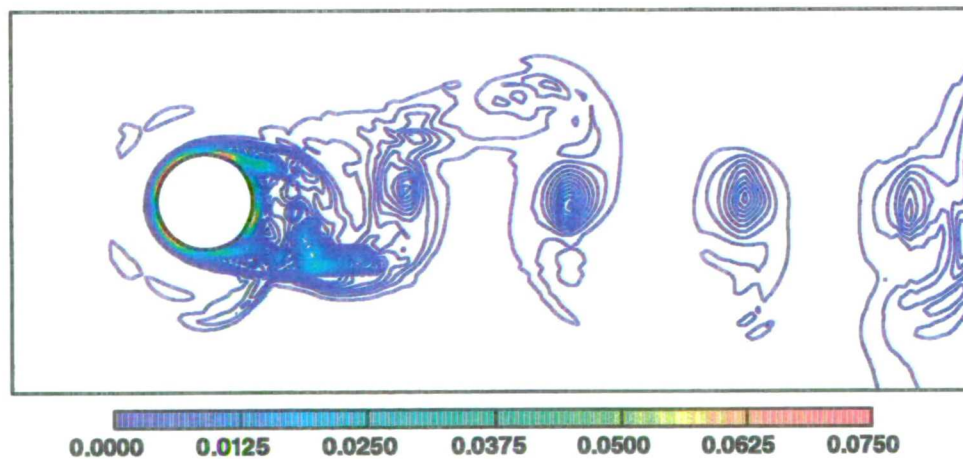


Figure 6.155: Vortex shedding: spatial Residual Error for EI on $Co = 0.4$.

The spatial part of the Residual Error estimate is shown in Fig. 6.155. It is very similar to the full Residual Error, as the bulk of the error comes from the spatial discretisation. The spatial error becomes much smaller in the vortex street

downstream of the cylinder.

The full Residual Error for the Backward Differencing in time and the Crank-Nicholson method look very similar. The real difference in the accuracy can be seen in the temporal part of the Residual Error estimate, shown in Figs. 6.156, 6.157 and 6.158. In the first instance, it can be seen that the temporal part of the residual highlights the different parts of the domain: the bulk of the error is located downstream of the cylinder, showing a distribution similar to the enstrophy field (see Fig. 6.152). The highest error is associated with the point of quickest variation of the velocity in time for this particular time-step. The temporal error is smallest for the Backward Differencing in time. One should, however, keep in mind that the temporal error changes in every time step and that it represents only about 10% of the total error.

In order to establish the dependence of the temporal error on the size of the time-step, the calculation is repeated with the Euler Implicit temporal discretisation and a maximum Courant number of 2.

The distribution of the full and temporal Residual Error estimates is shown in Figs. 6.159 and 6.160 for the same time as in the previous calculation. The full Residual Error is now around 10 % higher. The largest change can be seen in the temporal error estimate, three times higher than for $Co = 0.4$. Having in mind that the inlet velocity is set to $U_0 = 1 \text{ m/s}$, both the total error and its temporal part do not seem to be very high, giving the maximum error of 8% and 2%, respectively.

However, the influence of this error is not as small as it seems: the temporal error shown in Fig. 6.160 represents the distortion of the velocity field caused by temporal discretisation in a single time-step. A pressure cycle with the maximum Courant number of 2 is in this test case resolved over 40 time-steps, pushing the potentially cumulative temporal error much higher.

The effect of the higher Courant number can be clearly seen in the pressure trace for the two calculations, Fig. 6.161. The amplitude of the pressure oscillation is now much lower and the shedding frequency is underpredicted. For good temporal accuracy, it is therefore essential to keep the Courant number at a reasonable level,

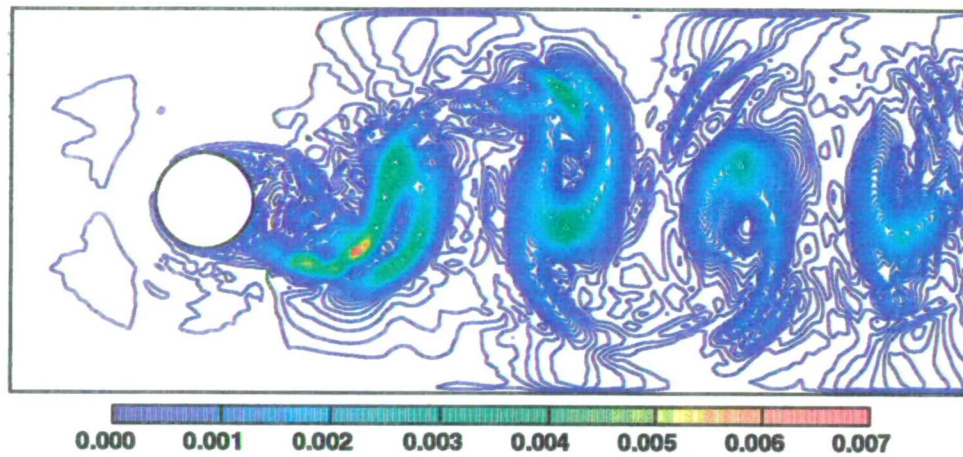


Figure 6.156: Vortex shedding: temporal Residual Error for EI on $Co = 0.4$.

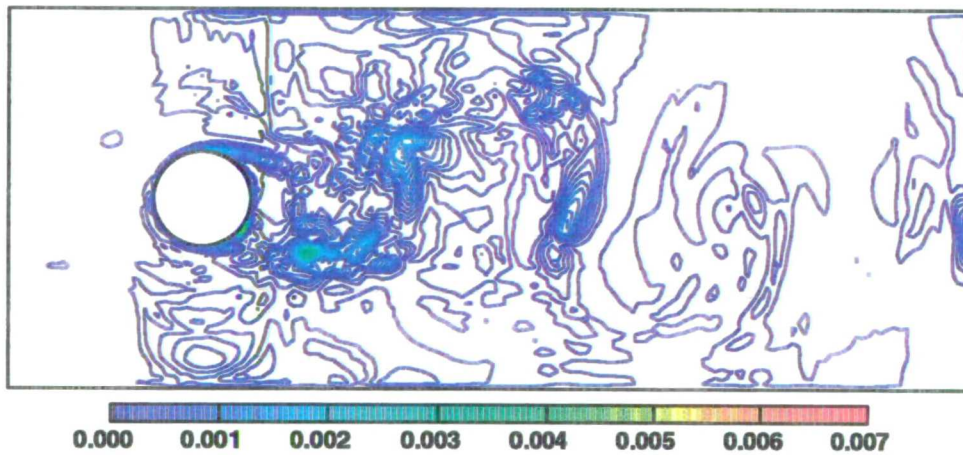


Figure 6.157: Vortex shedding: temporal Residual Error for BD on $Co = 0.4$.

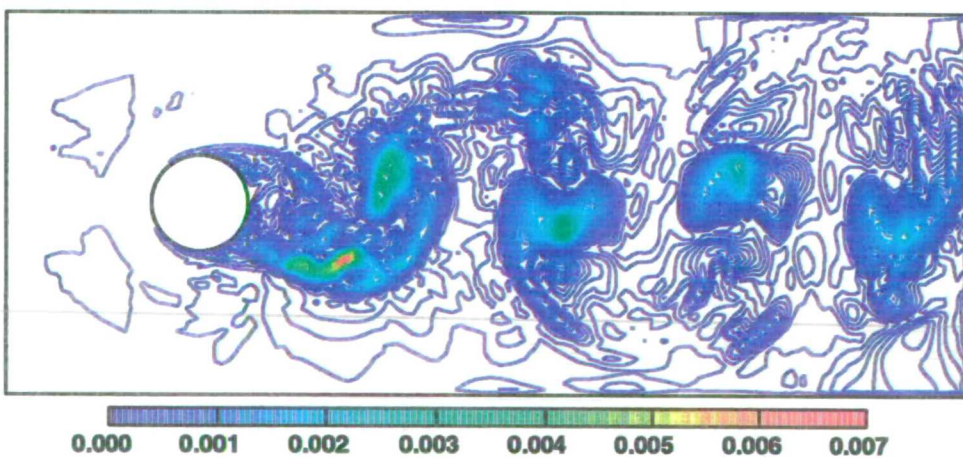


Figure 6.158: Vortex shedding: temporal Residual Error for CN on $Co = 2$.

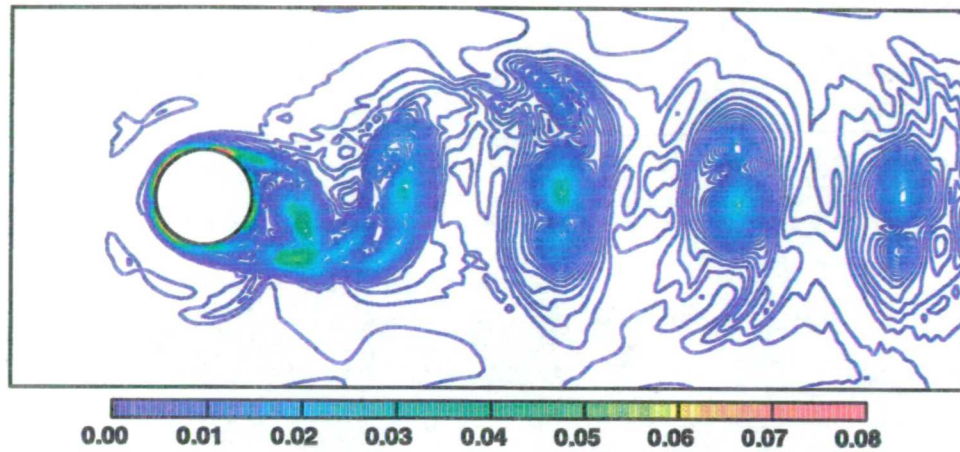


Figure 6.159: Vortex shedding: full Residual Error for EI on $Co = 2$.

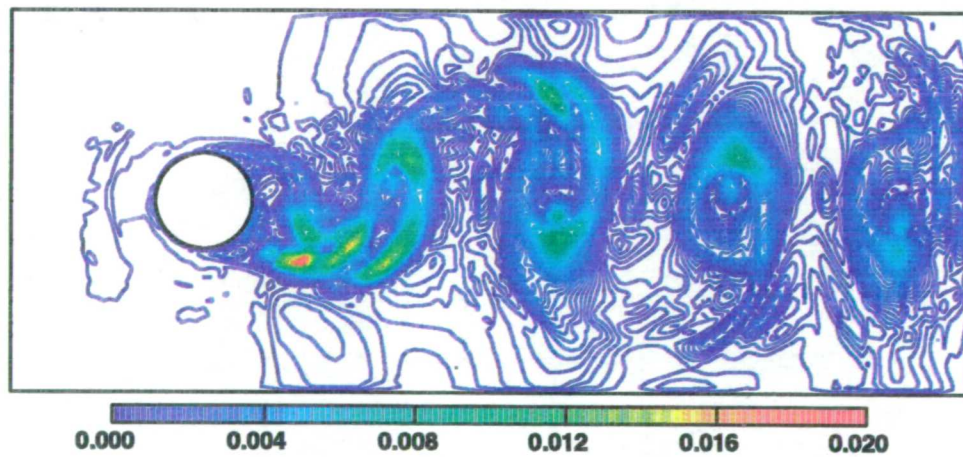


Figure 6.160: Vortex shedding: temporal Residual Error for EI on $Co = 2$.

depending on the size of the phase error that can be tolerated.

Local mesh refinement in transient calculations can be used in two ways. The first, tracking of the flow features through the domain, is interesting in cases of flame propagation and transient supersonic flows. The adaptive refinement is used to provide good mesh resolution around the moving feature, removing the cells when they are no longer needed. The proposed local refinement algorithm is not well-suited for such calculations, as it does not guarantee the recovery of the original mesh after refinement and unrefinement.

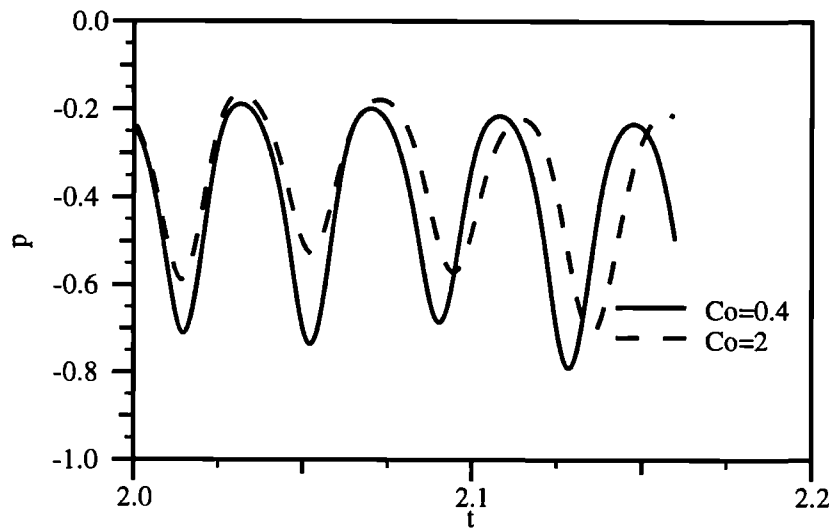


Figure 6.161: Vortex shedding: pressure trace for the Euler Implicit discretisation on two Courant numbers.

In periodic transient calculations, the same principle can be used. The refinement/unrefinement algorithm would now keep on working on the same parts of the domain during the shedding cycle. However, the price of refinement and field transfer between the meshes makes this approach relatively expensive, as the refinement needs to be repeated every several time-steps. An alternative to this treatment would be to follow the calculation through one cycle, estimating the error and marking the cells for refinement after every time-step. When the cycle is completed, the marked cells are refined, creating a mesh which can be kept unchanged during the calculation and still offers good resolution where it is needed.

An example of the above strategy on the vortex shedding test case will now be presented. Fig. 6.162 shows the example mesh that was created by refining all the cells with high error in a single time step (the velocity and pressure fields for the same time-step can be seen in Figs. 6.150 and 6.151). The mesh refinement covers

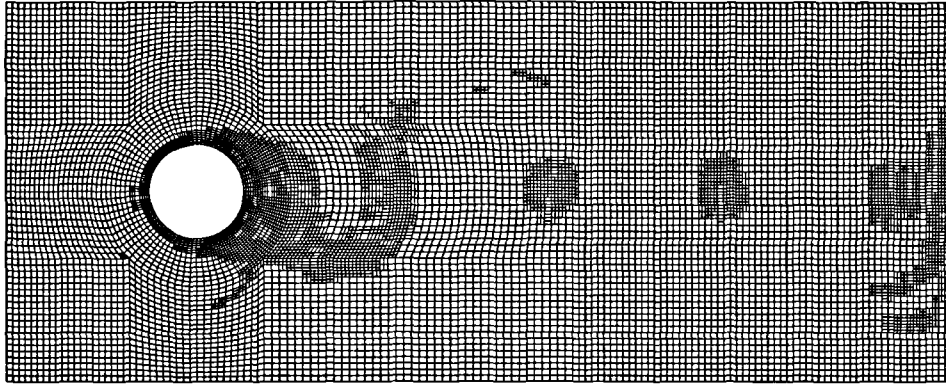


Figure 6.162: Vortex shedding: adaptively refined mesh changing in time.

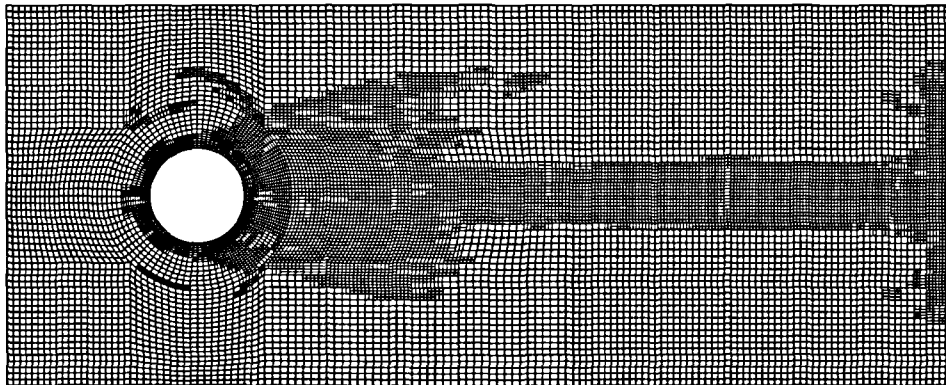


Figure 6.163: Vortex shedding: mesh refinement based on the error in the complete shedding cycle.

the regions of high velocity gradients and rapid pressure variation in the vortex street. As the calculation progresses in time, the desired effect of mesh removal has not been obtained. Only a very small number of cells close to the cylinder is unrefined, as the local velocity gradients are still relatively high. The refinement eventually covers much of the domain close to the cylinder, similar to the mesh shown in Fig. 6.163.

The second refinement strategy uses the refinement information accumulated

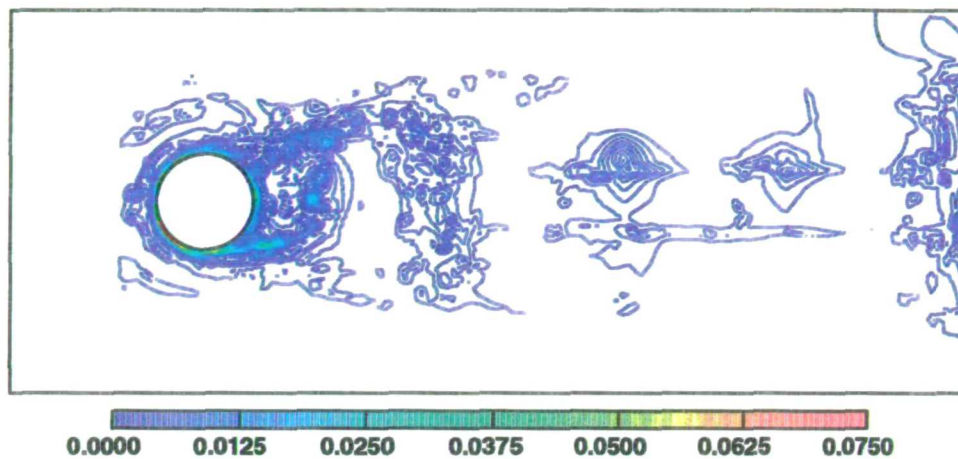


Figure 6.164: Vortex shedding: spatial Residual Error after the first level of refinement.

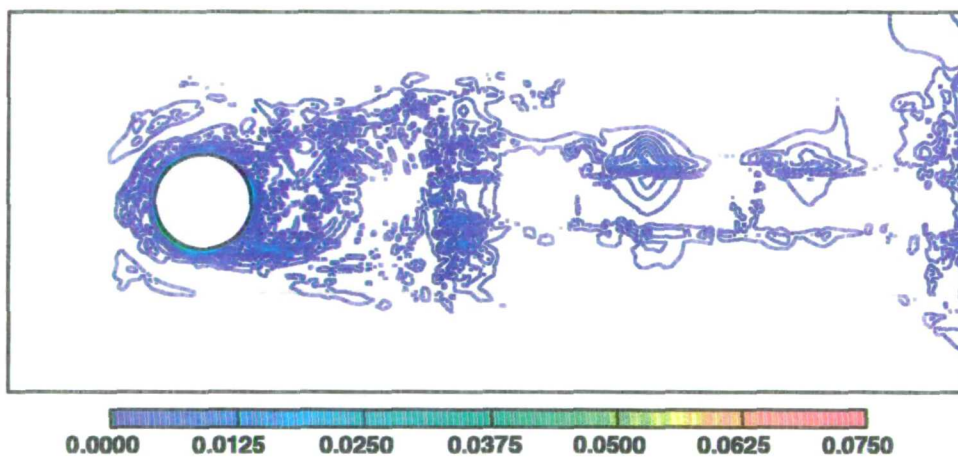


Figure 6.165: Vortex shedding: spatial Residual Error after the second level of refinement.

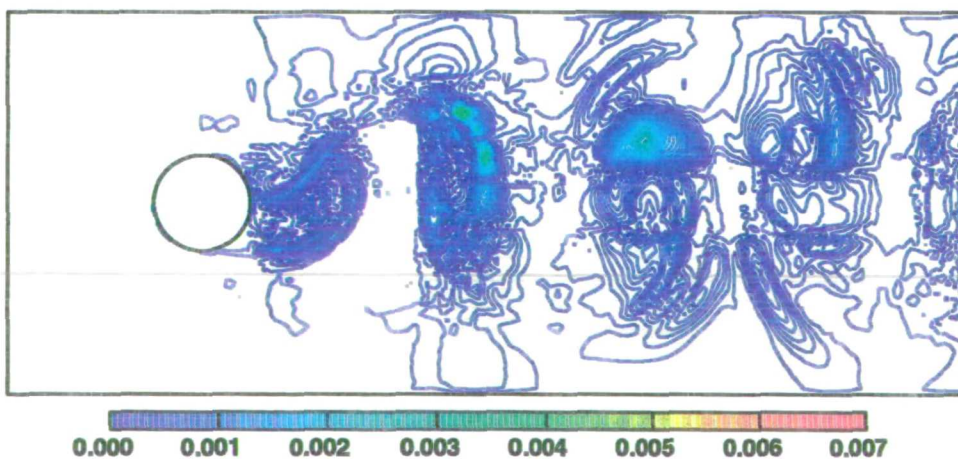


Figure 6.166: Vortex shedding: temporal Residual Error after the second level of refinement.

during the complete shedding cycle. It produces the refined mesh shown in Fig. 6.163. Refinement covers the surface of the cylinder, extends downstream over the part of the domain where the vortices are created and follows them to the outlet. It also covers a considerable part of the outlet boundary. Here, the high error is caused by the interaction of the boundary condition and the velocity field. The zero gradient condition used on this boundary is not appropriate when a vortex exits the domain. This causes a distortion of the velocity field and high local errors.

The influence of local mesh refinement on the accuracy of the solution can be seen in the distribution of the spatial part of the Residual Error estimate after one and two refinement levels, shown in Figs. 6.164 and 6.165. The size of the time-step is kept the same as on the initial mesh, increasing the maximum Courant number to 1.6 after two refinement levels. The error peaks close to the wall are now removed. The distribution of the temporal error is also different: the maximum error has moved outside of the refined region (see Fig. 6.163). The overall solution accuracy is now better, as a consequence of the better resolution in the area where the vortices are created.

6.6 Closure

This Chapter presented five example applications of the error estimates and adaptive mesh refinement. The variety of the presented flow regimes allows us to draw several conclusions on the overall performance of the proposed error-controlled local mesh refinement algorithm.

In supersonic flows, the solution in large parts of the domain varies very smoothly, with the narrow regions of high gradients. In such situations, the local mesh refinement algorithm is at its best. The bulk of the error is caused by inappropriate mesh resolution around the shocks and contact discontinuities. The task of error estimation is comparatively easy – an error indicator is completely appropriate for this purpose. From the point of view of adaptivity, the refinement/unrefinement procedure behaves better than refinement-only. The capability of cell removal al-

allows the adaptive calculation to be started from a comparatively fine mesh, which is able to pick up all the details of the flow. Mesh unrefinement consequently removes the cells where they are not needed.

Laminar calculations present a large contrast to the supersonic flows. The flow field here is very smooth, thus presenting a greater challenge for both error estimation and adaptive refinement. It is now possible to create a fine-mesh solution and use it to examine the accuracy of the error estimates. The Moment and Residual Error estimates bound the exact error, while the Direct Taylor Series Error estimate underpredicts its value. The performance of the adaptive refinement technique is limited by the quality of the mesh created by cell addition. In laminar calculations, only a part of the solution error is caused by insufficient mesh resolution, which is the type of the error removed by mesh refinement. As the number of cells increases, the importance of mesh quality becomes larger. The proposed refinement procedure does not produce the meshes of appropriate quality, which slows down the error reduction rate.

The two-dimensional turbulent test case reveals a variety of problems related to turbulent flows. The reduction error of the maximum error with the increase in the mesh size is very slow, primarily as a consequence of the complexity of the mathematical model. The main source of the problem is the near-wall region, for both wall-function treatment and the integration to the wall with the low- Re models. The use of wall-functions is connected with a limitation on the size of the near-wall cell. When the error is trapped near the wall, it cannot be further decreased. This is true for both uniform and adaptive mesh refinement, finally resulting in the inability to create mesh-independent solutions. The low- Re near-wall treatment does not pose such limitations, allowing the use of mesh adaptivity to its full extent. In this case, however the details of the flow near the wall need to be fully resolved, considerably increasing the overall computational cost.

The refinement criterion in turbulent calculations cannot be based on the error estimate for only one of the fields. Typically, each of the variables varies rapidly in different parts of the domain, which needs to be taken into account when the cells

for refinement are selected.

In spite of the low error reduction rate, the adaptive algorithm in turbulent calculations still offers the benefits in terms of accuracy, primarily by providing good mesh resolution at a relatively low cost.

A three-dimensional flow over the swept backward-facing step illustrates the behaviour of the adaptive algorithms in 3-D. The number of cells needed to resolve a flow of such complexity is very high. In the situations of highly complex flow fields and limited computer resources, the error-controlled local mesh refinement algorithm is a very useful tool, enabling a solution of reasonable accuracy to be obtained on a relatively small number of cells.

The final test case illustrates the application of the error estimate on a transient flow problem. The vortex shedding behind the cylinder gives us an opportunity to examine the relative influence of the spatial and temporal error on the overall solution accuracy. The transient formulation of the Residual Error estimate reveals that the temporal component of the error for a single time-step is considerably smaller than its spatial counterpart. The potentially cumulative effect of this error is still important, as it influences the shedding frequency. The overall accuracy of the transient flow simulation depends on both the spatial and temporal component of the error. Their balance should be estimated having in mind the number of time-steps that are needed to resolve the evolution of the flow in time. The local mesh refinement algorithm in this case can be used to eliminate the spatial component of the error. In order to reduce the cost of the adaptive calculation, this has been done by marking the cells for refinement through the whole shedding cycle and then using the accumulated information to refine the mesh.

This completes the assessment of the error estimation and local mesh refinement algorithms presented in this Thesis. In the next Chapter, the main findings of the Thesis are summarised and some recommendations are given for future research.

Chapter 7

Summary and Conclusions

The main concern of this study is the accuracy of numerical simulations of fluid flows with the Finite Volume method of discretisation. Several important areas of interest have arisen. Primarily, it is necessary to understand the assumptions introduced during the discretisation process and their implications on the quality of the solution. When these implications are recognised, it is easier to use the discretisation method in a such a way to produce accurate results with a minimum expense in terms of time and computer resources.

The desired numerical accuracy depends on the objective of the analysis and the quality of the applied mathematical models. The error caused by the discretisation should be considerably smaller than the expected modelling error. If the results of the simulations are to be used with confidence, it is necessary to provide an estimate of the discretisation error in absolute terms.

It is often known in advance what level of discretisation error can be accepted. An automatic procedure which creates a numerical solution of pre-determined accuracy would therefore be a useful extension of the error estimation tools. This can be achieved through an error-controlled adaptive mesh refinement procedure, in which the the mesh is dynamically changed depending on the estimated error levels. Three primary areas of research can therefore be distinguished: discretisation, error estimation and adaptive mesh refinement.

7.1 Discretisation

The Finite Volume method is a well-established numerical procedure, particularly suitable for fluid flow simulations because of its conservative properties. The discretisation method described in this study splits the computational domain into polyhedral control volumes with a variable number of faces. The governing equations of continuum mechanics are consequently discretised in the integral form over each control volume. The discretisation is performed in real space, using a fixed Cartesian coordinate system on meshes that do not change in time. The discretisation uses a colocated variable arrangement, with all fields sharing the same control volumes.

This study presents a formal analysis of the accuracy of the Finite Volume method in the second-order framework. Certain modifications introduced during the discretisation result in discretisation errors, influencing the order of accuracy of the method. The error terms are divided into numerical diffusion and the mesh-induced errors. The numerical diffusion tensors resulting from the convection term and temporal derivative are derived and their characteristics are illustrated on simple situations. The influence of mesh skewness and non-orthogonality on the accuracy is also examined.

Having recognised the convection discretisation as the largest source of numerical diffusion, Section 3.4 describes a new bounded and second-order accurate differencing scheme suitable for arbitrarily unstructured meshes. The Gamma differencing scheme described in this Section is a bounded version of Central Differencing. It belongs to the family of the Normalised Variable differencing schemes, first proposed by Leonard [82] and Gaskell and Lau [49]. A modification in the formulation of the Normalised Variable needed to use the boundedness criterion on non-uniform arbitrarily unstructured meshes has also been presented. It is now possible to calculate the value of the Normalised variable on a face-by-face basis, independent of the mesh spacing and without the explicit reference to the far upwind node. The resulting differencing scheme uses blending and switching between Upwind and Central

Differencing, with a very low level of numerical diffusion. The Gamma differencing scheme, however, shares to some extent the convergence problems encountered in most schemes of the NVD family.

The Finite Volume discretisation described above has been applied to fluid flow problems. The solution algorithm for the Navier-Stokes system in the segregated framework with and without the turbulence model is summarised. The pressure-velocity coupling is treated by the SIMPLE algorithm (Patankar[105]) for steady-state calculations. For transient problems, the PISO algorithm by Issa [66] has been used.

7.2 Error Estimation

The problem of *a-posteriori* error estimation for steady-state calculations has been approached in three different ways, resulting in three new error estimates presented in Chapter 4.

The Direct Taylor Series Error estimate originates from the Taylor series truncation error analysis. The discretisation error is calculated from the leading term of the truncation error, similar to the widely accepted Richardson extrapolation approach. However, unlike Richardson extrapolation, the Direct Taylor Series Error estimate calculates the error from a single numerical solution, using the Gauss' theorem twice to obtain the necessary second gradient tensor.

The differential equation that is being solved can be used to derive the transport equations for higher moments of the variable. If the original transport equation were solved exactly, all higher moment equations would also be satisfied. The numerical solution is, however, of limited accuracy and this is not the case. The Moment Error estimate uses the imbalance in the higher moment transport equations in order to estimate the discretisation error. The imbalance is calculated on a cell-by-cell basis and normalised to provide a measure of the absolute error level.

The derivation of the Residual Error estimate comes from the analysis of the basic assumptions in the Finite Volume discretisation. In order to calculate the

surface and volume integrals in the integral form of the governing equation, it is necessary to prescribe the variation of the solution over the control volume. At the same time, it is also necessary to determine the face value of the dependent variable from the distribution around the face. In the framework of the second-order accurate Finite Volume method both of these functions are linear, but they offer two different values of the variable for any point inside the control volume. The inconsistency between the two values is used to assemble the cell residual. This is a function which measures how well the solution satisfies the original equation over the control volume and is therefore naturally associated with the discretisation error. A suitable normalisation practice is again used to estimate the error magnitude, thus creating the Residual Error estimate.

A comparison of different methods of temporal discretisation allows extension of the formulation of the residual and consequently the Residual Error estimate to transient calculations. It is also possible to separate the parts of the total error caused by the temporal and spatial discretisation, which in turn can be used to adjust the size of the time-step to provide equivalent spatial and temporal accuracy.

The formulation of the residual proposed above allows the strict upper bound on the error in the energy norm to be obtained in the manner suggested by Ainsworth and Oden in [1, 2, 3, 4] for the Finite Element method. The Local Problem Error estimate is modified in a way appropriate for the Finite Volume discretisation and accompanied with the necessary error flux balancing procedure. In order to calculate the global error norm, an elliptic local error problem needs to be solved over every control volume. A simplified solution procedure for the local problem is also suggested.

The proposed error estimates are tested on several test cases with analytical solutions. The Moment and Residual Error estimate consistently show good performance with the Direct Taylor Series estimate underpredicting the error levels as expected.

7.3 Adaptive Mesh Refinement

The distribution of the estimated error is used to improve the quality of consequent solutions by modifying the computational mesh. This is done through an automatic local mesh refinement algorithm described in Chapter 5. The local refinement procedure used in this study consists of two parts: embedded cell-based h -refinement and a new unrefinement method based on cell pairs.

The refinement algorithm allows us to add more computational points in the parts of the domain where more resolution is needed. The algorithm marks the cells in the original mesh based on the pre-defined error threshold and propagates the refinement patterns through the mesh in order to enforce “1-irregularity”. This condition preserves the quality of the refined mesh by enforcing a smooth transition between the coarse and fine regions. The direction of the cell split is determined from the gradient of the solution, thus allowing the refinement procedure to create refinement patterns aligned with the flow. The new cells are embedded into the original mesh and are treated fully implicitly. This treatment allows a creation of an adaptive solution procedure without violating the conservative properties of the discretisation. The cell-based refinement is capable of creating rapid mesh grading with a large number of embedded levels of refinement.

The primary requirement on the unrefinement algorithm was to allow the removal of not only the cells added during refinement, but also of the cells from the initial mesh. An unrefinement procedure satisfying this requirement is original to this work. It merges the cell marked for unrefinement with one of its neighbours (also marked for unrefinement) to create a larger cell. Smooth mesh grading is again enforced through “1-irregularity”. This method of unrefinement is cheaper and faster than the traditional approach, which stores the history of refinement into a “tree” data-structure and is capable of removing only the cells added in previous refinement levels. The main drawback of the unrefinement algorithm in its current form is the interlocking of unrefinement patterns, which prevents further unrefinement and increases mesh non-orthogonality. It is also difficult to ensure that the original mesh

will be recovered when the cells added by refinement need to be removed.

The performance of the adaptive algorithm is examined on a two-dimensional case with an analytical solution, in conjunction with the exact error and the error estimates described above.

7.4 Performance of the Error-Controlled Adaptive Refinement Algorithm

The Finite Volume discretisation, error estimation and local mesh refinement are the components of an error-controlled adaptive refinement algorithm, capable of automatically producing a solution of some pre-determined accuracy. Before such a tool can be used for engineering purposes, it is necessary to examine the limits of its applicability and its performance on the variety of flow problems. For this purpose, five test cases have been examined, covering laminar and turbulent, incompressible and supersonic flows in two and three spatial dimensions. The conclusions drawn from these test cases can be summarised as follows:

- The adaptive refinement algorithm is at its best when the error caused by insufficient mesh resolution covers only a small portion of the computational domain. In this case, local refinement efficiently removes the error by augmenting mesh resolution in the affected areas. The error estimation in such cases is also relatively simple, as it is only necessary to highlight the points of high error. A typical example of such a situation is the supersonic flow with strong shocks.
- Laminar flows present the exact opposite of the situation described above. The error in the smoothly changing solution is caused not only by the insufficient mesh resolution, but also by the quality of the mesh. The error estimation now performs very well, with the Maximum and Residual Error estimates bounding the exact error. Local mesh refinement, however, does not offer great gains because of the deteriorating mesh quality. If the refinement is limited to a

modest number of levels, the gains from the improved mesh resolution are still greater than the drawbacks of lower mesh quality.

- Turbulent flows present a considerable challenge for the local refinement algorithm. It has been noted that the drop in the maximum error is much slower than expected, both for uniform and adaptive refinement. The main source of difficulty is the near-wall region. If this region is bridged by the wall-functions, the size of the near-wall cell is limited, thus preventing further refinement near the wall. If a low- Re turbulence model is used instead, a very large number of cells is needed to resolve the rapid variation of the solution near the wall. It is no longer possible to use the error estimate on only one of the fields to judge the accuracy of the solution and decide on mesh refinement. This needs to be taken into account in adaptive calculations by consulting the error estimate for more than one variable. In spite of the slow error reduction, local mesh refinement still offers some benefits, mainly through good mesh resolution in the regions of high gradients and efficient use of the available computer resources.
- In transient calculations, the overall accuracy of the solution depends on both the spatial and temporal components of the error. The transient formulation of the Residual Error estimate allows discrimination between the two. Although the proposed local mesh refinement algorithm is not well suited for the transient calculations with the tracking of flow features, it can still be successfully used on periodic flow problems.

The present study contributes to the field of Computational Fluid Dynamics in three ways. In the area of Finite Volume discretisation, the Gamma differencing scheme offers a capability of improved accuracy, by creating bounded solutions with a minimum amount of numerical diffusion. The proposed error estimates present a reliable tool for predicting the levels of discretisation errors in a wide variety of flow regimes. Finally, the properties of the local mesh refinement algorithm are examined on a variety of flow problems and some conclusions on its performance are given.

7.5 Future Work

The results of this study can be improved in several ways, mainly in the areas of discretisation accuracy and local mesh refinement. The following recommendations for future work are proposed.

The critical part of the Finite Volume method from the point of view of solution accuracy is the discretisation of the convection term. An accurate convection differencing scheme needs to preserve the boundedness of the solution and at the same time introduce as little numerical diffusion as possible. At this stage, there is no obvious alternative to the non-linear switching/blending schemes of the TVD and NVD family. Scope for improvement within the existing framework still exists, particularly from the point of view of the convergence of the scheme in steady-state calculations.

Several researchers (*eg.* Choi *et al.* [30]) have recently noted that the performance of the TVD and NVD schemes deteriorates on highly non-orthogonal meshes. The discretised form of the convection term does not immediately reveal the cause of this behaviour. Numerical experiments, however, suggest that mesh non-orthogonality needs to be taken into account in some way. In spite of the fact that high mesh non-orthogonality is regularly present in industrial calculations, no attempts in this direction have been reported to date.

The field of error estimation in the Finite Volume method is still in its early stages of development. In general, room for improvement of the accuracy of error estimates always exists. The primary objective in the near future will be extensive validation of the proposed error estimates on a much larger number of test cases, having in mind the potential of their further improvement. The main uncertainty at this stage lies in the normalisation of cell imbalances. A more appropriate formulation can be obtained only when the error estimates have been applied on a variety test cases, well beyond the scope of this Thesis.

In the author's opinion, the most important issue in adaptive mesh refinement is the development of an automatic mesh generation tool capable of producing high-

quality meshes. In recent years, several commercial automatic mesh generators have emerged, but none of them is yet capable of providing the mesh quality needed for accurate CFD calculations. Some of the characteristics of a good mesh generator would be the following:

- The mesh generator should automatically create a computational mesh with a given number of cells in an arbitrarily complex three-dimensional domain described by a CAD package.
- The mesh should consist of mainly hexahedral control volumes with low level of distortion and non-orthogonality, particularly in the near-wall region. The mesh generator needs to be able to create smooth grading towards some pre-defined points, with the capability of mesh alignment with a prescribed direction.
- It should be possible to automatically re-create the mesh in order to modify the cell size in different parts of the domain, based on the information obtained from an error estimate. At the same time, the mesh generator should interact with the CAD model, such that the details of boundary description are not lost in the refinement process.

Once such a tool is available, it will replace the proposed local mesh refinement algorithm. In the meantime, the existing algorithm can be improved in several ways, mainly in its unrefinement part. Firstly, it is necessary to develop a strategy which will prevent the inter-locking of the cells. Although the problem looks simple, it requires some sort of global optimisation or pattern recognition logic.

If the refinement/unrefinement procedure is to be used in transient tracking calculations, a further modification is needed. The proposed method of point addition and removal does not guarantee that the original mesh will be retrieved after multiple levels of refinement and unrefinement. This needs to be enforced by storing the cell pairs which should be preferentially merged when the cells are due for unrefinement. With such a modification, the current form of the unrefinement algorithm

would be capable of recovering the original mesh and would avoid the inter-locking problem.

Finally, the overall error-controlled local refinement procedure should be subjected to further testing in order to establish the influence of the different refinement parameters on the quality of results, particularly in turbulent flow regimes.

Appendix A

Comparison of the Euler Implicit Discretisation and Backward Differencing in Time

In this Appendix it will be shown that the difference between the Backward Differencing in time and the Euler Implicit temporal discretisation (see Section 3.3.2) cancels the D_C and D_D terms in the temporal error, Eqs. (3.108) and (3.109). For details of the derivation of the numerical diffusion term for the Euler Implicit temporal discretisation, the reader is referred to Section 3.6.

The Euler Implicit temporal discretisation gives the following discretised form of the temporal derivative, Eqn. (3.38):

$$\frac{\partial \phi}{\partial t} = \frac{\phi^n - \phi^o}{\Delta t}.$$

The Backward Differencing in time, on the other hand, uses Eqn. (3.54):

$$\frac{\partial \phi}{\partial t} = \frac{\frac{3}{2}\phi^n - 2\phi^o + \frac{1}{2}\phi^{oo}}{\Delta t}.$$

When the difference between the two terms is integrated over the control volume and the time-step, using the assumption of linear variation of ϕ in space, the following

term is obtained:

$$\begin{aligned}
 E_{BD-EI} &= \int_t^{t+\Delta t} \int_{V_P} \rho \left[\left(\frac{\partial \phi}{\partial t} \right)_{BD} - \left(\frac{\partial \phi}{\partial t} \right)_{EI} \right] dt dV \\
 &= \rho_P V_P \frac{\phi^n - 2\phi^o + \phi^{oo}}{2 \Delta t} \\
 &= \frac{1}{2} \rho_P V_P \Delta t \frac{\phi^n - 2\phi^o + \phi^{oo}}{\Delta t^2} \\
 &= \frac{1}{2} \rho_P V_P \Delta t \frac{\partial^2 \phi}{\partial t^2}. \tag{A.1}
 \end{aligned}$$

The second derivative of ϕ in time in Eqn. (A.1) will be obtained by differentiating Eqn. (3.111):

$$\frac{\partial(\rho\phi)}{\partial t} = Su + Sp\phi - \nabla \cdot (\rho \mathbf{U} \phi) + \nabla \cdot (\rho \Gamma_\phi \nabla \phi).$$

Assuming that ρ , $\mathbf{U}$, Γ_ϕ , Su and Sp are constant:

$$\rho \frac{\partial^2 \phi}{\partial t^2} = Sp \left(\frac{\partial \phi}{\partial t} \right) - \nabla \cdot (\rho \mathbf{U} \frac{\partial \phi}{\partial t}) + \nabla \cdot \left[\rho \Gamma_\phi \nabla \left(\frac{\partial \phi}{\partial t} \right) \right]. \tag{A.2}$$

Finally, combining Eqs. (A.1) and (A.2) and discretising the convection and diffusion terms, it follows:

$$E_{BD-EI} = -\frac{\Delta t}{2} \sum_f F \left(\frac{\partial \phi}{\partial t} \right)_f + \frac{\Delta t}{2} \sum_f (\rho \Gamma_\phi)_f \mathbf{S} \cdot \frac{\partial(\nabla \phi)_f}{\partial t} + \frac{\Delta t}{2} Sp V_P \left(\frac{\partial \phi}{\partial t} \right)_P, \tag{A.3}$$

which, using Eqs. (3.108), (3.109) and (3.110) can be written as:

$$E_{BD-EI} = D_C + D_D - D_S. \tag{A.4}$$

Section 3.6 shows that the difference between the Crank-Nicholson method and Euler Implicit temporal discretisation can be written as:

$$E_t = D_C + D_D + D_S. \tag{A.5}$$

The convection and diffusion errors D_C and D_S therefore cancel out, leaving the source term error D_S , which is a consequence of the source term linearisation.

Bibliography

- [1] Ainsworth, M.: “*A-posteriori* error estimators for second-order elliptic systems: Part 1. Theoretical foundations and *a-posteriori* error analysis”, *Computers Math. Applic.*, 25(2):101–113, 1993.
- [2] Ainsworth, M. and Oden, J. T.: “A procedure for *a-posteriori* error estimation for $h - p$ Finite Element methods”, *Comp. Meth. Appl. Mech. Engineering*, 101:73–96, 1992.
- [3] Ainsworth, M. and Oden, J. T.: “*A-posteriori* error estimators for second-order elliptic systems: Part 2. An optimal order process for calculating self-equilibrating fluxes”, *Computers Math. Applic.*, 26(9):75–87, 1993.
- [4] Ainsworth, M. and Oden, J.T.: “A unified approach to *a-posteriori* error estimation using Element Residual methods”, *Numerische Mathematik*, 65:23–50, 1993.
- [5] Almeida, G.P., Durao, D.F.G., and Heitor, M.V.: “Wake flows behind two-dimensional model hills”, *Experimental Thermal and Fluid Science*, 7:87–101, 1993.
- [6] Anderson, D.A.: “Adaptive grid methods for partial differential equations”, In Ghia, K.N. and Ghia, U., editors, *Advances in grid generation*. ASME, New York, 1983.
- [7] Aris, R.: *Vectors, tensors and the basic equations of fluid mechanics*: Dover Publications, 1989.

- [8] Babuška, I. and Rheinboldt, W.C.: "*A-posteriori* error estimates for the Finite Element method", *Int. J. Comp. Meth. Engineering*, 12:1597 1615, 1978.
- [9] Babuška, I. and Rheinboldt, W.C.: "Error estimates for adaptive Finite Element calculations", *SIAM J. Numer. Analysis*, 15(4):736 753, 1978.
- [10] Babuška, I. and Rheinboldt, W.C.: "Reliable error estimation and mesh adaptation for the Finite Element method", In J.T., Oden, editor, *Computational Methods in Nonlinear Mechanics*, pages 67–108. North Holland, New York, 1980.
- [11] Bank, R.E.: "*A-posteriori* error estimates. Adaptive local mesh refinement and multigrid iteration", In *Lecture notes in mathematics, multigrid methods II*, pages 7 22. Springer Verlag, 1986.
- [12] Bank, R.E. and Weiser, A.: "Some *a-posteriori* error estimators for elliptic partial differential equations", *Math. Comput.*, 44:283 301, 1985.
- [13] Beam, R.M. and Warming, R.F.: "An implicit Finite-Difference algorithm for hyperbolic system in conservation law form", *J. Comp. Physics*, 22:87 109, 1976.
- [14] Beam, R.M. and Warming, R.F.: "An implicit factored scheme for the compressible Navier-Stokes equations", *AIAA Journal*, 16:393 402, 1978.
- [15] Berge, M.J.: *Adaptive mesh refinement for hyperbolic partial differential equations*, PhD thesis, Department of Computer Science, Stanford University, 1982.
- [16] Berger M J and Collela, P.: "Local adaptive mesh refinement for shock hydrodynamics", *J. Comp. Physics*, 82:64 84, 1989.
- [17] Berger, M. J. and Oliger, J.: "Adaptive mesh refinement for hyperbolic partial differential equations", *J. Comp. Physics*, 53:484 512, 1984.

- [18] Berger, M.J.: "On conservation at grid interfaces", *SIAM J. Numer. Analysis*, 24(5):967-984, 1987.
- [19] Berger, M.J. and Jameson, A.: "An adaptive multigrid method for the Euler equations", *Lecture Notes in Physics*, 218:92-97, 1985.
- [20] Boris, J.P. and Book, D.L.: "Flux-corrected transport. I. SHASTA, a fluid transport algorithm that works", *J. Comp. Physics*, 11:38-69, 1973.
- [21] Boussinesq, J.: "Théorie de l'écoulement tourbillant", *Mem. par. div. Sav., Paris*, 23, 1877.
- [22] Brandt, A.: "Multi-level adaptive solutions to boundary-value problems", *Math. Comp.*, 31:333, 1977.
- [23] Caretto, L., Curr, R.M., and Spalding, D.B.: "Two numerical methods for three-dimensional boundary layers", *Comp. Methods Appl. Mech. Eng.*, 1:39, 1972.
- [24] Caruso, S.: *Adaptive grid techniques for fluid flow problems*, PhD thesis, Thermosciences Division, Department of Mechanical Engineering, Stanford University, 1985.
- [25] Caruso, S., Ferziger, J.H., and Oliger, J.: "Adaptive grid techniques for elliptic flow problems", Technical Report TF-23, Thermosc. Div. Stanford University, 1985.
- [26] Cebeci, T. and Smith, A.M.O.: *Analysis of turbulent boundary layers*: Academic Press, 1974.
- [27] Chakravarthy, S.R. and Osher, S.: "High resolution application of the OSHER upwind scheme for the Euler equation": AIAA Paper 83-1943, 1983.
- [28] Chen, W.L., Lien, F.S., and Leschziner, M.Z.: "A local grid refinement scheme within a multiblock structured-grid strategy for general flows", In *6th Int. Symp. on CFD*, Lake Tahoe, USA, September 1995.

- [29] Chien, K.Y.: 'Predictions of channel and boundary-layer flows with a low-Reynolds-number turbulence model', *AIAA Journal*, 20:33–38, 1982.
- [30] Choi, S.K., Nam, H.Y., and Cho, M.: "A comparison of higher-order bounded convection schemes", *Comp. Meth. Appl. Mech. Engineering*, 121:281–301, 1995.
- [31] Coelho, P., Pereira, J.C.F., and Carvalho, M.G.: "Calculation of laminar recirculating flows using a local non-staggered grid refinement system", *Int. J. Num. Meth. Fluids*, 12(6):535–557, 1991.
- [32] Courant, R., Friedrichs, K.O., and Lewy, H.: "Über die partiellen differenzgleichungen der mathematischen Physik", *Mathematische Annalen*, 100:32–74, 1928.
- [33] Courant, R., Isaacson, E., and Rees, M.: "On the solution of non-linear hyperbolic differential equation by finite differences", *Comm. Pure Appl. Math.*, 5:243, 1952.
- [34] Dafa'Alla, A.D., Juntasaro, E., and Gibson, M.M.: "Calculation of oscillating boundary layers with the $q - \zeta$ turbulence model", In *3rd International Symposium on Engineering Turbulence Modelling and Measurements, Crete, Greece*, 1996.
- [35] Dandekar, H.W., Hlavacek, V., and J., Degreve.: "An explicit 3D Finite-Volume method for simulation of reactive flows using a hybrid moving adaptive grid", *Numerical Heat Transfer, part B*, 24(1):1–29, 1993.
- [36] Darwish, M.S.: 'A new high-resolution scheme based on the normalized variable formulation', *Numerical Heat Transfer, part B* 24:353–371 1993.
- [37] Darwish, M.S. and Moukalled, F.: "Normalized variable and space formulation methodology for high-resolution schemes", *Numerical Heat Transfer part B* 26 1–79 96, 1994.

- [38] Deardorff, J.W.: "A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers", *J. Fluid Mechanics*, 41(2):453–480, 1970.
- [39] Demirdžić, I., Perić, M., and Lilek, Ž: "A colocated finite volume method for predicting flows at all speeds", *Int. J. Num. Meth. Fluids*, 16:1029–1050, 1993.
- [40] Demkowicz, L., Oden, J.T., Rachowicz, W., and Hardy, O.: "Toward a universal $h-p$ adaptive Finite Element strategy: Part 1: Constrained approximation and data structure", *Comp. Meth. Appl. Mech. Engineering*, 77:79–112, 1989.
- [41] Dwyer, H.A.: "Grid adaptation for problems in fluid dynamics", *AIAA Journal*, 22(12):1705–1712, December 1984.
- [42] Emery, A.F.: "An evaluation of several differencing methods for inviscid fluid flow problems", *J. Comp. Physics*, 2:306–331, 1968.
- [43] "ERCOFTAC Workshop on Data Bases and testing of calculation Methods for Turbulent Flows": 4th ERCOFTAC/IAHR Workshop of Refined Flow Modelling, Karlsruhe, Germany, April 1995.
- [44] Eswaran, V. and Pope, S.B.: "Direct numerical simulations of the turbulent mixing of a passive scalar", *Physics of Fluids*, 31(3):506–520, 1988.
- [45] Eswaran, V. and Pope, S.B.: "An examination of forcing in direct numerical simulations of turbulence", *Computers and Fluids*, 16(3):257–278, 1988.
- [46] Favre, A.: "Equations des gaz turbulents compressibles", *J. de Mécanique*, 4:361, 1965.
- [47] Ferziger, J.H.: "Simulation of incompressible turbulent flows", *J. Comp. Physics*, 69:1–48, 1987.
- [48] Ferziger, J.H. and Perić, M.: *Computational methods for fluid dynamics*: Springer Verlag, Berlin-New York, 1995.

- [49] Gaskell, P.H. and Lau, A.K.C.: "Curvature-compensated convective transport: SMART, a new boundedness-preserving transport algorithm", *Int. J. Num. Meth. Fluids*, 8:617-641, 1988.
- [50] Gentry, R.A., Martin, R.E., and B.J., Daly: "An Eulerian differencing method for unsteady compressible flow problems", *J. Comp. Physics*, 1:87, 1966.
- [51] Gibson, M.M. and Dafa'Alla, A.D.: "Two-equation model for turbulent wall flow", *AIAA Journal*, 33(8):1514-1518, 1995.
- [52] Girault, V. and Raviart, P.-A.: *Finite Element methods for Navier-Stokes equations*, volume 5 of *Springer series in computational mathematics*: Springer-Verlag, 1986.
- [53] Givi, P.: "Model-free simulations of turbulent reactive flows", *Prog. Energy Combust. Sci.*, 15:1-107, 1989.
- [54] Gosman, A.D. and Ideriah, H.J.K.: "TEACH-2e: A general computer program for two-dimensional turbulent recirculating flows", Technical report, Fluids Section, Department of Mechanical Engineering, Imperial College, London, 1983.
- [55] Gosman, A.D. and Lai, K.Y.: "Finite Difference and other approximations for the transport and Navier-Stokes equations", In *Proc IAHR Symposium on Refined Modelling of Fluid Flows*, 1982.
- [56] Hanjalić, K. and Launder, B.E.: "A Reynolds stress model of turbulence and its application to thin shear layer flows", *J. Fluid Mechanics*, 52:609, 1972.
- 57 Harlow, F.H. and Nakayama, P.: 'Transport of turbulence energy decay rate', Technical Report Report LA 3854, Los Alamos National Laboratory, 1968
- [58] Harten, A.: "High resolution schemes for hyperbolic conservation laws", *J. Comp. Physics*, 49:357-393, 1983.

- [59] Harten, A.: "On a class of high resolution total variation stable Finite Difference schemes", *SIAM J. Numer. Analysis*, 31:1 23, 1984.
- [60] Hawken, D.F., Gottlieb, J.J., and Hansen, J.S.: "Review of some adaptive node-movement techniques in Finite-Element and Finite-Difference solutions of partial differential equation", *J. Comp. Physics*, 95:254 302, 1991.
- [61] Haworth, D C., El Tahry, S.H., and Huebler, M.S.: "A global approach to error estimation and physical diagnostics in multidimensional fluid dynamics", *Int. J. Numer. Meth. Fluids*, 17(1):75 97, 1993.
- [62] Helf, C. and Küster, U.: "A Finite Volume method with arbitrary polygonal control volumes and high order reconstruction for the Euler equations", In Wagner, S., Hirschel, E.H., Périaux, J., and Piva, R., editors, *Computational Fluid Dynamics*, pages 234 238. J. Wiley and Sons, September 1994.
- [63] Hestens, M.R. and Steifel, E.L.: "Method of conjugate gradients for solving linear systems", *Journal of Research*, 29:409 436, 1952.
- [64] Hinze, J.O.: *Turbulence*: McGraw-Hill, 1975.
- [65] Hirsch, C.: *Numerical computation of internal and external flows*: John Willey & Sons, 1991.
- [66] Issa, R.I.: "Solution of the implicitly discretized fluid flow equations by operator-splitting", *J. Comp. Physics*, 62:40 65, 1986.
- [67] Jacobs, D.A.H.: "Preconditioned Conjugate Gradient methods for solving systems of algebraic equations": Central Electricity Research Laboratories, 1980.
- [68] Jameson, A., Schmidt, W., and Turkel, E.: "Numerical simulation of the Euler equations by Finite Volume methods using Runge-Kutta time stepping schemes": AIAA Paper 81-1259, 1981.

- [69] Jasak, H. and Weller, H.G.: "Interface-tracking capabilities of the InterGamma differencing scheme", Internal Report, CFD research group, Imperial College, London, February 1995.
- [70] Kallinderis, Y. and Vidwans, A.: "Generic parallel adaptive-grid Navier-Stokes algorithm", *AIAA Journal*, 32 1):54 61, 1994.
- [71] Kelly, D.W.: "The self-equilibration of residuals and complementary a posteriori error estimates in the Finite Element method", *Int. J. Num. Meth. Engineering*, 20:1491 1506, 1984.
- [72] Kern, T.: *Fehlerkontrolle für hyperbolische Differentialgleichungen und Navier-Stokes Gleichungen*, PhD thesis, Mathematischen Fakultät der Albert-Ludwigs- Universität Freiburg i.Br., 1993.
- [73] Khosla, P.K. and Rubin, S.G.: "A diagonally dominant second-order accurate implicit scheme", *Computers and Fluids*, 2:207 209, 1974.
- [74] Lam, C.K.G. and Bremhorst, K.A.: "Modified form of the $k - \epsilon$ model for predicting wall turbulence", *J. Fluids Engineering*, 103:456 460, 1981.
- [75] Launder, B.E., Reece, G.J., and Rodi, W.: "Progress in the development of a Reynolds-stress turbulence closure", *J. Fluid Mechanics*, 68(3):537 566, 1975.
- [76] Launder, B.E. and Sharma, B.I.: "Application of the energy-dissipation model of turbulence to the calculation of a flow near a spinning disk", *Letters in Heat and Mass Transfer*, 1:131 138, 1974.
- [77] Launder, B.E. and Spalding, D.B.: "The numerical computation of turbulent flows", *Comp. Meth. Appl. Mech. Engineering*, 3:269 289, 1974.
- [78] Lax, P.D.: "Weak solutions of non-linear hyperbolic equations and their numerical computation", *Comm. Pure Appl. Math* , 7:159 193, 1954
- [79] Lax, P.D. and Wendroff, B.: "Svstems of conservation laws" *Comm Pu e Appl. Math.*, 13:217 237, 1960.

- [80] LeHuuNho, E. and Béguier, C.: "Experimental study of the flow over a swept backward-facing step", to be submitted to the AIAA Journal.
- [81] Leonard, B.P.: "A stable and accurate convective modelling procedure based on quadratic upstream interpolation", *Comp. Meth. Appl. Mech. Engineering*, 19:59–98, 1979.
- [82] Leonard, B.P.: "Simple high-accuracy resolution program for convective modeling of discontinuities", *Int. J. Num. Meth. Fluids*, 8:1291–1318, 1988.
- [83] Leonard, B.P.: "The ULTIMATE conservative difference scheme applied to unsteady one-dimensional advection", *Comp. Meth. Appl. Mech. Engineering*, 88:17–74, 1991.
- [84] Lerat, A.: "Implicit methods of second order accuracy for Euler equations", *AIAA Journal*, 23:33–40, 1983.
- [85] Lerat, A. and Peyret, R.: "Non-centered schemes and shock propagation problems", *Computers and Fluids*, 2:35–52, 1974.
- [86] Leschziner, M.Z.: Private communication, 1995.
- [87] Lien, F.S. and Leschziner, M.A.: "Upstream monotonic interpolation for scalar transport with application to complex turbulent flows", *Int. J. Num. Meth. Fluids*, 19:527–548, 1994.
- [88] Lilek, Ž. and Perić, M.: "A fourth-order Finite Volume method with colocated variable arrangement", *Computers and Fluids*, 24(3):239–252, 1995.
- [89] Löhner, R.: "An adaptive Finite Element scheme for transient problems in CFD", *Comp. Meth. Appl. Mech. Engineering*, 61:323–338, 1987.
- [90] MacCormack, R.W.: "The effect of viscosity in hypervelocity impact cratering": AIAA Paper 69-354, 1969.
- [91] MacCormack, R.W.: "A numerical method for solving the equations of compressible viscous flow": AIAA Paper 81-0110, 1981.

- [92] McGuirk, J.J. and Rodi, W.: "A depth-averaged mathematical model for the near field of side discharges into open channel flow", *J. Fluid Mechanics*, 86:761, 1978.
- [93] McGuirk, J.J., Taylor, A.M.P.K., and Whitelaw, J.H.: "The assessment of numerical diffusion in the upwind-difference calculations of turbulent recirculating flows", In Bradbury, L.J.S., Durst, F., Launder, B.E., Schmidt, F.W., and Whitelaw, J.H., editors, *Selected papers from the Third International Symposium on Turbulent Shear Flows*, volume 3 of *Turbulent Shear Flows*, pages 206–224, The University of California, Davis, September 9–11 1981.
- [94] Métais, O. and Lesieur, M.: "Spectral large-eddy simulation of isotropic and stably stratified turbulence", *J. Fluid Mech.*, 239:157–194, 1992.
- [95] Moin, P. and Kim, J.: "Numerical investigation of turbulent channel flow", *J. Fluid Mechanics*, 118:341–377, 1982.
- [96] Moinat, P. and Lesieur, M.: "Large-eddy simulations of turbulent flows over a swept backward-facing step", Technical report, Laboratoire des Ecoulements Géophysiques et Industriels - Institut de Mécanique de Grenoble, 1995.
- [97] Muzaferija, S.: *Adaptive Finite Volume method for flow prediction using unstructured meshes and multigrid approach*, PhD thesis, Imperial College, University of London, 1994.
- [98] Oden, J.T., Demkowicz, L., Rachowicz, W., and Westermann, T.A.: "Toward a universal $h - p$ adaptive Finite Element strategy: Part 2: A posteriori error estimation", *Comp. Meth. Appl. Mech. Engineering*, 77:113–180, 1989.
- [99] Oden, J.T., Demkowicz, L., Strubolis, T., and Devloo, P.: "Adaptive methods for problems in solid and fluid mechanics", In Babuška, I., Zienkiewicz, O.C., de Gago, S.R., and de Oliveira, A., editors, *Adaptive Methods and Error Refinement in Finite Element Calculations*. John Wiley & Sons 1986.

- [100] Oden, J.T., Stroubolis, T., and Devloo, P.: "Adaptive Finite Element methods for the analysis of inviscid compressible flow: Part 1: Fast refinement/unrefinement and moving mesh methods for unstructured meshes", *Comp. Meth. Appl. Mech. Engineering*, 59:327 362, 1986.
- [101] Oden, J.T., Wu, W., and Ainsworth, M.: "An *a-posteriori* error estimate for Finite Element approximations of the Navier-Stokes equations", *Comp. Meth. Appl. Mech. Engineering*, 111:185 202, 1994.
- [102] Oden, J.T., Wu, W., and Legat, V.: "An $h - p$ adaptive strategy for Finite Element approximations of the Navier-Stokes equations", *Int. J. Num. Meth. Fluids*, 20:831-851, 1995.
- [103] Osher, S. and Chakravarthy, S.R.: "High resolution schemes and the entropy condition", *SIAM J. Numer. Analysis*, 21(5):955 984, 1984.
- [104] Panton, R.L.: *Incompressible flow*: John Wiley and Sons, 1984.
- [105] Patankar, S.V.: *Numerical heat transfer and fluid flow*: Hemisphere Publishing Corporation, 1981.
- [106] Patankar, S.V. and Baliga, B.R.: "A new Finite-Difference scheme for parabolic differential equations", *Numerical Heat Transfer*, 1:27, 1978.
- [107] Patankar, S.V. and Spalding, D.B.: "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows", *Int. J. Heat Mass Transfer*, 15:1787, 1972.
- [108] Patel, V.C., Rodi, W., and Scheurer, G.: "Turbulence models for near-wall and low Reynolds number flows: a review", *AIAA Journal*, 23:1308 1319, 1985.
- [109] Perić, M.: *A Finite Volume method for the prediction of three-dimensional fluid flow in complex ducts*, PhD thesis, Imperial College, University of London, 1985.

- [110] Phillips, R.E: *A multilevel multigrid method for recirculating flows*, PhD thesis, Pennsylvania State University, 1984.
- [111] Rachowicz, W., Oden, J.T., and Demkowicz, L.: "Toward a universal $h - p$ adaptive Finite Element strategy: Part 3: Design of $h - p$ meshes", *Comp. Meth. Appl. Mech. Engineering*, 77:181 212, 1989.
- [112] Raithby, G.D.: "A critical evaluation of upstream differencing applied to problems involving fluid flow", *Comp. Meth. Appl. Mech. Engineering*, 9:75 103, 1976.
- [113] Raithby, G.D.: "Skew-upstream differencing schemes for problems involving fluid flow", *Comp. Meth. Appl. Mech. Engineering*, 9:153 164, 1976.
- [114] Raithby, G.D. and Torrance, K.E.: "Upstream-weighted schemes and their application to elliptic problems involving fluid flow", *Computers and Fluids*, 2:191 206, 1974.
- [115] Ramakrishnan, R.: "Structured and unstructured grid adaptation schemes for numerical modelling of field problems", *Appl. Numer. Mathematics*, 14(1-3):285 310, 1994.
- [116] Ramakrishnan, R., Bey, K.S., and Thornton, E.A.: "Adaptive quadrilateral and triangular Finite Element scheme for compressible flows", *AIAA Journal*, 28 1):51 59, January 1990.
- [117] Rhie, C.M. and Chow, W.L.: "A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation": AIAA-82-0998, AIAA/ASME 3rd Joint Thermophysics, Fluids, Plasma and Heat Transfer Conference, St. Louis, Missouri, 1982.
- [118] Roe, P.L.: "Large scale computations in fluid mechanics, Part 2", In *Lectures in Applied Mathematics*, volume 22, pages 163 193. Springer Verlag, 1985.

- [119] Rogallo, R.S. and Moin, P.: "Numerical simulation of turbulent flows", *Annual Review of Fluid Mechanics*, 16:99–137, 1984.
- [120] Rogers, M.M. and Moin, P.: "The structure of the vorticity field in homogeneous turbulent flows", *J. Fluid Mechanics*, 176:33–66, March 1987.
- [121] Rotta, J.C.: "Statistische Theorie nichthomogener Turbulenz", *Z. Phys.*, 129:547, 1951.
- [122] Sharif, M.A.R. and Busnaina, A.A.: "Evaluation and comparison of bounding techniques for convection-diffusion problems", *J. Fluids Engineering-Transactions of the ASME*, 115(1):33–40, 1993.
- [123] Shih, T.H., Liou, W.W, Shabir, A., Yang, Z., and Zhu, J.: "A new $k - \epsilon$ eddy viscosity model for high Reynolds number turbulent flows": NASA TM 106721, 1994.
- [124] Simpson, R.B.: "Automatic local refinement for irregular rectangular meshes", Technical report, Department of Computer Science, University of Waterloo, 1978.
- [125] Sonar, T., Hannemann, V., and Hempel, D.: "Dynamic adaptivity and residual control in unsteady compressible flow computation", *Mathematical and Computer Modelling*, 20(10-11):201–213, 1994.
- [126] Spalding, D.B.: "A novel Finite-Difference formulation for differential expression involving both first and second derivatives", *Int. J. Comp. Meth. Engineering*, 4:551, 1972.
- [127] Speziale, C.G.: "On non-linear $k - l$ and $k - \epsilon$ models of turbulence", *J. Fluid Mechanics*, 178:459–475, 1987.
- [128] "STAR-CD Version 2.1 - Manuals": Computational Dynamics Limited, 1991.
- [129] Sweby, P.K.: "High resolution schemes using flux limiters for hyperbolic conservation laws", *SIAM J. Numer. Analysis*, 21:995–1011, 1984.

- [130] Tattersall, P. and McGuirk, J.J.: "Evaluation of numerical diffusion effects in viscous flow calculations", *Computers and Fluids*, 23(1):177–209, 1994.
- [131] Temam, R.: *Navier-Stokes equations: Theory and numerical analysis*: North-Holland, Amsterdam, 2nd edition, 1985.
- [132] Tennekes, H. and J.L., Lumley: *A first course in turbulence*: The MIT Press, 1972.
- [133] Thompson, J.F.: "A survey of dynamically-adaptive grids in the numerical solution of partial differential equations", *Appl. Numer. Mathematics*, 1(1):3–27, 1985.
- [134] Thompson, M.C. and Ferziger, J.H.: "An adaptive multigrid technique for the incompressible Navier-Stokes equations", *J. Comp. Physics*, 82:94–121, 1989.
- [135] Uslu, S.: *Numerical prediction of transonic flow in turbine blade passages*, PhD thesis, Imperial College, University of London, 1994.
- [136] Van Der Vorst, H.A.: "Bi-CGSTAB: A fast and smoothly converging variant of Bi-CG for the solution of nonsymmetric linear systems", *SIAM J. Scientific Computing*, 13(2):631–644, 1992.
- [137] Van Doormaal, J.P. and Raithby, G.D.: "An evaluation of the segregated approach for predicting incompressible fluid flows", In *National Heat Transfer Conference, Denver, Colorado, August 4-7 1985*.
- [138] Van Leer, B.: "Towards the ultimate conservative differencing scheme", In Cabannes, H. and Temam, R., editors, *Proceedings of the Third International Conference on Numerical Methods in Fluid Mechanics*, volume 1, pages 163–168. Springer, 1973.
- [139] Van Leer, B.: "Towards the ultimate conservative differencing scheme. II Monotonicity and conservation combined in a second-order scheme", *J. Comp. Physics*, 14:361–370, 1974.

- [140] Van Leer, B.: "Towards the ultimate conservative differencing scheme. III. Upstream-centered finite-difference schemes for ideal compressible flow", *J. Comp. Physics*, 23:263–275, 1977.
- [141] Van Leer, B.: "Towards the ultimate conservative differencing scheme. IV. A new approach to numerical convection", *J. Comp. Physics*, 23:276–299, 1977.
- [142] Van Leer, B.: "Towards the ultimate conservative differencing scheme V: A second-order sequel to Godunov's method", *J. Comp. Physics*, 23:101–136, 1977.
- [143] Vanka, S.P.: "Block-implicit multigrid solution of Navier-Stokes equations in primitive variables", *J. Comp. Physics*, 65:138–158, 1986.
- [144] Vidwans, A. and Kallinderis, Y.: "A 3-D Finite-Volume scheme for the Euler equations on adaptive tetrahedral grids", *J. Comp. Physics*, 113(2):249–267, 1994.
- [145] Vilsmeier, R. and Hänel, D.: "Adaptive methods on unstructured grids for Euler and Navier-Stokes equations", *Computers and Fluids*, 22(4-5):485–499, 1993.
- [146] Warming, R.F. and Beam, R.M.: "Upwind second order difference schemes and applications in aerodynamic flows", *AIAA Journal*, 14:1241–1249, 1976.
- [147] Weller, H.G.: Private communication, 1993.
- [148] Wong, H.H. and Raithby, G.D.: "Improved Finite-Difference methods based on a critical evaluation of the approximating errors", *Numerical Heat Transfer*, 2:131, 1979.
- [149] Woodward, P. and Colella, P.: "The numerical simulation of two-dimensional fluid flow with strong shocks", *J. Comp. Physics*, 54:115–173, 1984.
- [150] Yakhot, V. and Orszag: "Renormalisation group analysis of turbulence. I. Basic theory", *SIAM J. Scientific Computing*, 1:3–51, 1986.

- [151] Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B., and Speziale, C.G.: “Development of turbulence models for shear flows by a double expansion technique”, *Physics of Fluids*, 4(7):1510–1520, July 1992.
- [152] Zalesak, S.T.: “Fully multidimensional flux-corrected transport algorithms for fluids”, *J. Comp. Physics*, 31:335–362, 1979.
- [153] Zhu, J.: “A low-diffusive and oscillation-free scheme”, *Communications in Applied Numerical Methods*, 7:225–232, 1991.
- [154] Zhu, J.: “On the higher-order bounded discretisation schemes for Finite Volume computations of incompressible flows”, *Comp. Methods Appl. Mech. Engineering*, 98:345–360, 1992.
- [155] Zhu, J. and Rodi, W.: “A low dispersion and bounded convection scheme”, *Comp. Methods Appl. Mech. Eng.*, 92:87–96, 1991.
- [156] Zienkiewicz, O. C. and Taylor, R. L.: *The Finite Element method, vol 1: Basic formulation and linear problems*: McGraw-Hill, 4th edition, 1989.