



CRANFIELD UNIVERSITY

COLLEGE OF AERONAUTICS

Department of Flow Control & Predictions

MSc THESIS

Academic Year 1996-7

E. M. Onomor

**Error Control in Finite Elements
for Fluid Modelling**

Supervisor : Professor A. J. Morris

October 1997

This thesis is submitted in partial fulfilment of the requirements for the degree of
Master of Science



CRANFIELD UNIVERSITY

COLLEGE OF AERONAUTICS

Department of Flow Control & Predictions

MSc THESIS

Academic Year 1996-7

E. M. Onomor

**Error Control in Finite Elements
for Fluid Modelling**

Supervisor : Professor A. J. Morris

October 1997

This thesis is submitted in partial fulfilment of the requirements for the degree of
Master of Science

ABSTRACT

The main objective of this thesis was to ascertain and to catalogue the possible sources of errors in the finite element method when used for fluid modelling. Some of these errors are inherent in the finite element method whilst others are introduced by the user when performing a fluid mechanical analysis. This thesis distinguishes from the two and provides where possible, ways of preventing or controlling them.

The form is based on the SAFESA Technical manual, which is an equivalent for structural analyses.

Focus is then made, as an example, on a real world problem; "The Backward Facing Step" where, based on experimental data an element sensitivity analysis is performed within ANSYS/FLOTTRAN. This analysis attempts to determine the magnitude of errors associated with prescribed element distortions in relation to aspect ratio, skew angle and taper ratio.

The results contained within are in the form of graphical velocity profile plots, contour plots and tables at the varying distortions. The results show a strong deviation from the control mesh as the distortion is increased.

ACKNOWLEDGEMENTS

I would like to thank my supervisor, Professor A. J. Morris for his suggestions and assistance, Dr R.Vignjevic for accepting to take over on short notice when my supervisor was not available and to Dr . Hemp for his patience and time.

I would also like to thank M. Navratnum, M. S. Manusia and A. O'Shea for assisting in the final construction of this thesis.

ABSTRACT	I
ACKNOWLEDGEMENTS	II
TABLE OF CONTENTS	III
CHAPTER 1 INTRODUCTION	1
CHAPTER 2 ERROR ASSESSMENT IN FLUID MODELLING	4
2.1 Identification of Fluid Modelling Errors	6
2.1.1 Mathematical Model of the Fluid Structure	6
2.1.2 Domain Assumptions	7
2.1.3 Fluid Properties	8
2.1.4 Boundary Condition	8
2.2 Treatment of Modelling/Idealisation Errors	9
2.2.1 Experience Based Rules	10
2.2.2 Scoping Calculations	10
2.2.3 Comparison with Test Results	11
2.2.4 Hierarchical Modelling	11
2.2.5 Sensitivity Analysis	11
2.3 Application of the Techniques	12
2.3.1 The Mathematical Model	12
2.3.2 The Domain	12
2.3.3 Fluid Properties	13
2.3.4 Boundary Conditions	14
CHAPTER 3 FLUID THEORY	15
3.1 FLUID FLOW FUNDAMENTALS	15
3.1.1 Continuity Equation	15
3.1.2 Momentum Equation	16
3.1.3 Energy Equation	18
3.2 Discretisation of Equations	20
3.2.1 Transient Term	21
3.2.2 Advection Term	22
3.2.3 Diffusion Terms	23
3.2.4 Source Terms	24
3.3 Solution Algorithm	24
3.3.1 Segregated Solution Algorithm	25

CHAPTER 4 BACKWARD FACING STEP	26
4.1 Model Definition	26
4.1.1 Domain Definition :	26
4.1.2 Definition of Boundary Conditions :	27
4.2 Anticipated Results :	28
4.3 Presented Results :	28
CHAPTER 5 SENSITIVITY ANALYSIS	29
5.1 Model Domain	29
5.2 PATRAN Model	29
5.3 Element FLUID141	31
5.3.1 Input Data	31
5.3.2 Output Data	32
5.3.3 Element Assumptions and Restrictions	32
5.3.4 Element Formulation Assumptions	32
5.4 Fluid Properties	33
5.5 Boundary Condition	33
5.6 Execution Control Parameters	34
5.6.1 Tri-diagonal Solver	34
5.6.2 Preconditioned Conjugate Matrix Solver	34
5.7 Derived Output Quantities	35
5.8 Model Validation	35
5.8.1 Mesh Independency	36
5.9 Quadrilateral Element Aspect Ratio	38
5.9.1 Methodology For Aspect Ratio Variation	38
5.9.2 Aspect Ratio Results	39
5.9.3 Element Aspect Ratio Gradient	41
5.10 Quadrilateral Element Skew	42
5.10.1 Methodology For Element Skew Variation	43
5.10.2. Element Skew Results	44
5.11 Quadrilateral Element Taper Ratio	46
5.11.1 Methodology For Taper Ratio Variation	46
5.11.2 Taper Results	47
5.12 Perturbed Element Downstream of Inspection Point	49

CONCLUSIONS	51
Further Work	51
REFERENCES	53
FIGURES	55
APPENDIX A	109

LIST OF FIGURES

Figure 1.1	Rib Geometry Simplification	55
Figure 1.2	Diffuser Boundary Symmetry	55
Figure 5.1	Dimensions of Analysis Model Domain	56
Figure 5.2	Definition of Surfaces within PATRAN	56
Figure 5.3	Model Boundary Conditions; Velocity and Pressure	57
Figure 5.4	Element Refinement Process from Model Mesh (1) to Model Mesh (3)	58
Figure 5.5	Mesh Density of Model Mesh (1)	58
Figure 5.6	Mesh Density of Model Mesh (2)	59
Figure 5.7	Mesh Density of Model Mesh (3)	59

Model Mesh (2) Results :

Figure 5.8	Velocity Profile at Three Mesh Densities at $x = 12$ mm	60
Figure 5.9	Velocity Profile at Three Mesh Densities at $x = 30$ mm	60
Figure 5.10	Contour Plot of Axial Velocity Variation	61
Figure 5.11	Contour Plot of Pressure Variation	61
Figure 5.12	Vector Plot of Velocity Variation	62
Figure 5.13	Magnification at Step Location	62
Figure 5.14	Contour Plot of Stream Function Variation; Averaged	63
Figure 5.15	Contour Plot of Stream Function Variation; Non-Averaged	63

Aspect Ratio Results :

Figure 5.16	Quad Aspect Ratio Definition	64
Figure 5.17	Mesh with Aspect Ratio = 2.4:1	65
Figure 5.18	Mesh with Aspect Ratio = 12:1	65
Figure 5.19	Aspect Ratio Variation to Inspection Point One	66
Figure 5.20	Magnification Plot at Step Location	66

Figure 5.21	Aspect Ratio Variation to Inspection Point Two	67
Figure 5.22	Magnification Plot at Channel Centre-line	68
Figure 5.23	Magnification Plot at Step Location	68

Aspect Ratio = 3.6:1

Figure 5.24	Contour Plot of Axial Velocity Variation	69
Figure 5.25	Contour Plot of Pressure Variation	69
Figure 5.26	Vector Plot of Velocity Variation	70
Figure 5.27	Magnification at Step Location	70
Figure 5.28	Contour Plot of Stream Function Variation; Averaged	71
Figure 5.29	Contour Plot of Stream Function Variation; Non-Averaged	71

Aspect Ratio = 7.2:1

Figure 5.30	Contour Plot of Axial Velocity Variation	72
Figure 5.31	Contour Plot of Pressure Variation	72
Figure 5.32	Vector Plot of Velocity Variation	73
Figure 5.33	Magnification at Step Location	73
Figure 5.34	Contour Plot of Stream Function Variation; Averaged	74
Figure 5.35	Contour Plot of Stream Function Variation; Non-Averaged	74

Aspect Ratio = 12:1

Figure 5.36	Contour Plot of Axial Velocity Variation	75
Figure 5.37	Contour Plot of Pressure Variation	75
Figure 5.38	Vector Plot of Velocity Variation	76
Figure 5.39	Magnification at Step Location	76
Figure 5.40	Contour Plot of Stream Function Variation; Averaged	77
Figure 5.41	Contour Plot of Stream Function Variation; Non-Averaged	77

Aspect Ratio Gradient Results :

Figure 5.42	Modified Mesh for Gradient Analysis	78
Figure 5.43	Steep Gradient Mesh	78
Figure 5.44	Medium Gradient Mesh	79
Figure 5.45	Smooth Gradient Mesh	79

Figure 5.46	Velocity Profile at Varying Aspect Ratio Gradient	80
Figure 5.47	Magnification Plot at Step Location	80

Skew Angle Results :

Figure 5.48	Quad Skew Angle Element Definition	81
Figure 5.49	Quad Skew Angle Element Normalisation Definition	81
Figure 5.50	Illustration of Element Skew Methodology	82
Figure 5.51	Mesh with Skew Angle = 50.2 degrees	83
Figure 5.52	Mesh with Skew Angle = 84.1 degrees	83
Figure 5.53	Skew Angle Variation to Inspection Point One	84
Figure 5.54	Magnification Plot at Step Location	84
Figure 5.55	Skew Angle Variation to Inspection Point Two	85
Figure 5.56	Magnification Plot at Channel Centre-line	86
Figure 5.57	Magnification Plot at Step Location	86

Skew Angle = 67.4 degrees

Figure 5.58	Contour Plot of Axial Velocity Variation	87
Figure 5.59	Contour Plot of Pressure Variation	87
Figure 5.60	Vector Plot of Velocity Variation	88
Figure 5.61	Magnification at Step Location	88
Figure 5.62	Contour Plot of Stream Function Variation; Averaged	89
Figure 5.63	Contour Plot of Stream Function Variation; Non-Averaged	89

Skew Angle = 78.2 degrees

Figure 5.64	Contour Plot of Axial Velocity Variation	90
Figure 5.65	Contour Plot of Pressure Variation	90
Figure 5.66	Vector Plot of Velocity Variation	91
Figure 5.67	Magnification at Step Location	91
Figure 5.68	Contour Plot of Stream Function Variation; Averaged	92
Figure 5.69	Contour Plot of Stream Function Variation; Non-Averaged	92

Skew Angle = 84.1 degrees

Figure 5.70	Contour Plot of Axial Velocity Variation	93
Figure 5.71	Contour Plot of Pressure Variation	93

Figure 5.72	Vector Plot of Velocity Variation	94
Figure 5.73	Magnification at Step Location	94
Figure 5.74	Contour Plot of Stream Function Variation; Averaged	95
Figure 5.75	Contour Plot of Stream Function Variation; Non-Averaged	95

Taper Ratio Results :

Figure 5.76	Quad Taper Ratio Variation	96
Figure 5.77	Illustration of Element Taper Methodology	97
Figure 5.78	Mesh with Element Taper = 0.5	98
Figure 5.79	Mesh with Element Taper = 0.2	98
Figure 5.80	Taper Ratio Variation to Inspection Point One	99
Figure 5.81	Magnification Plot at Step Location	99
Figure 5.82	Taper Ratio Variation to Inspection Point Two	100
Figure 5.83	Magnification Plot at Channel Centre-Line	101
Figure 5.84	Magnification Plot at Step Location	101

Taper Ratio = 0.33

Figure 5.85	Contour Plot of Axial Velocity Variation	102
Figure 5.86	Contour Plot of Pressure Variation	102
Figure 5.87	Vector Plot of Velocity Variation	103
Figure 5.88	Magnification at Step Location	103
Figure 5.89	Contour Plot of Stream Function Variation; Averaged	104
Figure 5.90	Contour Plot of Stream Function Variation; Non-Averaged	104

Taper Ratio = 0.2

Figure 5.91	Contour Plot of Axial Velocity Variation	105
Figure 5.92	Contour Plot of Pressure Variation	105
Figure 5.93	Vector Plot of Velocity Variation	106
Figure 5.94	Magnification at Step Location	106
Figure 5.95	Contour Plot of Stream Function Variation; Averaged	107
Figure 5.96	Contour Plot of Stream Function Variation; Non-Averaged	107
Figure 5.97	Perturbed Element Down Stream of Inspection Point Two	108
Figure 5.98	Magnification Plot at Step Location	108

Chapter 1 Introduction

“All the mathematical sciences are founded on relations between physical laws and laws of numbers, so that the aim of exact science is to reduce the problems of nature to the determination of quantities by operation with numbers.”

James Clerk Maxwell, 1856

Computational fluid dynamics (CFD) has unfurled over the forty years of its existence from a speciality of a small closely knit consortium of enthusiasts into a vast and multi-dimensional enterprise. The material resources committed to CFD research and development are still expanding as existing areas of application continue to increase and new areas are constantly emerging.

As a results of this expansion and proliferation is a CFD today has widened to incorporate a number of disciplines and specialisation ranging from aerodynamics to oil and gas reservoir exploitation and from shore protection to medical science.

There are four distinct streams of numerical solution techniques being used in CFD. These are finite difference, finite volume, spectral methods and finite element methods. In outline the numerical methods that form the basis of the solver perform the following steps:

- Approximation of the following flow variables by means of simple functions.
 - Discretisation by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
-

- Solutions of the algebraic equations.

The main differences between the four are related to the way in which the flow variables are approximated and with the discretisation processes.

Finite difference methods describe the unknowns ϕ of the flow problem by means of point samples at the node points of a grid of co-ordinate lines. Truncated Taylor series lines are often used to generate finite difference approximations of derivatives of ϕ in terms of point samples of ϕ at each grid point and its immediate neighbours. Those derivatives in the governing equation are replaced by finite differences yielding an algebraic equation for the values of ϕ at each grid point.

The finite volume method was originally developed as a special finite difference formulation. It involves a formal integration of the governing equations of the fluid flow over all the finite control volumes of the solution domain. Discretisation involves the substitution of a variety of finite difference type approximations for the terms in the integrated equation representing the flow processes such as convection, diffusion and sources. This converts the integral equations into a system of algebraic equations. These algebraic equations are then solved iteratively.

Spectral methods approximate the unknowns by means of truncated Fourier series or series of Chebyshev polynomials. These approximations are valid throughout the entire computational domain. Again we replace the unknowns in the governing equations by a truncated series. The constraints that leads to the algebraic equations for the Fourier or Chebyshev series is provided by a weighting function or by making the approximate function coincide with the exact solution at a number of grid points.

The finite element methods, use simple piecewise functions valid on elements to describe the local variations of unknown flow variables ϕ . The governing equation is precisely satisfied by the exact solution ϕ . If the piecewise approximating functions for ϕ are substituted into the equation, it will not hold exactly and a residual is defined to measure

the errors. The errors are then minimised by multiplying them by a set of weighting functions and integrating. As a consequence, a set of algebraic equations for the unknown coefficients of the approximating functions are obtained.

Associated with these methods, are errors that affect the general accuracy of the solution. It is the intention of the author to concentrate the work in this thesis on the types of errors related to the finite element method.

Chapter 2 Error Assessment in Fluid Modelling

Whether it be a structure or a fluid, the process of computer modelling and simulation will have inevitable associated errors. They will arise from various sources and because it is not possible to eliminate them, it is important to know how and where they arise.

Since the whole purpose of computer modelling is to simulate, as accurately as possible, the physical world, to have any level of confidence in the results one must be able to estimate these errors.

Errors should not be confused with uncertainties arising from our lack of understanding of the real world and its natural variability. Errors come about from the methods employed to describe the real world computationally whereas uncertainties are embodied in the description of the real world.

To be sure of accounting for all the possible sources of error, the analyst should follow a standard procedure. All too often, though, much reliance is placed in the expertise of the user to recognise modelling errors. Though certainly invaluable, it is not enough to make an informed judgement. One way a user may recognise an error is by the presence of inconsistencies in the solution. Though this is a valid indicator an absence of inconsistencies does not automatically ensure precise results.

It is possible to identify four main classes of errors in finite element analysis :-

1. modelling errors - from the idealisation* process
2. procedural errors - from discretisation, meshing and subsequent post-processing
3. formulation errors - associated with solving the finite element model
4. system software errors - typical of any computer system.

For the modelling of structures there exists a procedure called the SAFESA (Safe Structural Analysis) method. It was developed to provide a disciplined approach when applying finite element analysis to structural modelling and in doing so, minimise the opportunity for error. SAFESA mainly concerns itself with the treatment of modelling or idealisation errors. It does this by using the estimates of errors, which the analyst will have to make, together with their sources to refine the idealisation process until the errors are acceptably small. This is discussed in more detail in section 2.2. For full details of this procedure, the reader is referred to the SAFESA technical manual [1].

No such method exists for fluid modelling, but it is the intention of the following section to use SAFESA as a basis for identifying or classifying potential fluid modelling errors.

*idealisation provides the basis of the computer model which is generated by discretisation and meshing.

2.1 Identification of Fluid Modelling Errors

The reader will recall in the previous section that errors in the application of finite element analysis can be divide into four main classes. These, however, give no indication of the source of errors or allow their effects to be assessed using SAFESA. The main idealisation/ modelling error sources are :

- mathematical model of the structure
- domain assumptions
- fluid properties
- boundary conditions and loading.

These apply to fluid modelling in the following way.

2.1.1 Mathematical Model of the Fluid Structure

The analyst has to select a mathematical model to describe the physical situation that is being simulated. To do this means making assumptions on how the fluid behaves. This undoubtedly introduces approximations and associated errors. But these behavioural assumptions are needed so that the physical laws can be manipulated mathematically to give a useful set of expressions. Some examples of assumptions :

- flow dimension, i.e. 2D or 3D
- state of flow, i.e. laminar or turbulent
- idealisation of fluid, i.e. compressible or incompressible.

2.1.2 Domain Assumptions

It is often the case that an analysis is performed only on a section of an entire system. For example, the cooling flow in the nozzle guide vane of a gas turbine. In such circumstances the analyst has to make a judgement on the limits of the domain i.e. where to draw the boundary between the domain and the system. Ideally one would extend the domain as far downstream from the point of interest as possible to allow the flow to stabilise. Often, though, it is constrained by physical limits and/or whether the boundary conditions are known.

With a nozzle guide vane the domain could consist only of the blade or it could include the disc as well. Not extending the domain far enough may mean that features crucial to the analysis may be idealised as a boundary condition which will inevitably lead to errors - domain errors.

Of course if the eliminated portion is symmetrical to the portion being analysed then there is no error introduced. In the case of modelling the flow in a diffuser, as shown in Figure 1.1, only half of the cross sectional area need be meshed.

This is not the only cause of domain error. Another possible originator is the simplification of the physical structure over/through which the fluid flows. For example, it is common practice to add ribs in the cooling passages of turbine vanes to promote fluid turbulence which increases the rate of heat transfer. The fluid flow is very much dependent on the shape of the rib so any simplifications in modelling the ribs, as in Figure 1.2, could produce errors.

Another example is the smoothing out of any small irregularities on the surface. In the physical world, however, these irregularities are liable to affect the value of surface roughness which in turn can have a direct bearing on the value of Reynolds number.

However, in all cases, it will be the responsibility of the analyst to make a judgement on how accurate the structure will have to be modelled. In some cases simplification may not cause significant changes in the fluids behaviour. If the irregularity or disturbance is over a relatively short distance compared to the total distance considered, or if it does not protrude far into the flow stream, it is conceivable that even in reality it will not have any effect. So it would be a trade off between the time needed for accurate modelling and the significance of simplification errors.

2.1.3 Fluid Properties

In any analysis the analyst will have to make assumptions on the values of the different properties. In the case of fluids, values for viscosity and density, will be a direct consequence of what assumptions have been made on the fluid behaviour, i.e. laminar or turbulent flow. The wrong fluid properties will lead to significant errors in the modelling solution.

2.1.4 Boundary Condition

The boundary is the interface between the domain and the entire system. It is important to know, as accurately as possible, the conditions at this interface. Taking the NGV example it would mean knowing the pressure and temperature of the air entering the blade, and knowing the conditions of the main gas stream at the point that the coolant flow is ejected from the blade. The analyst would also have to know the blade metal temperature to assess the amount of heat pickup by the coolant air, which will affect its physical properties. Knowing the true values of these parameters is, however, not possible so consequently assumptions will have to be made which brings associated errors.

In the case of flow in a pipe, an example of a boundary error could be the assumption of a uniform velocity profile. Likewise, if modelling the flow from the exit of a gas turbine combustor, it would be incorrect to assume a uniform temperature and pressure profile.

Another cause of boundary errors is due to the way in which the boundary conditions are imposed as a result of the fluid model. However, it is the authors opinion that this is not significant in fluid modelling since, unlike with structural modelling, it is not possible to mix elements i.e. 3D section in a 2D system.

2.2 Treatment of Modelling/Idealisation Errors

Having covered the different sources of errors this section will discuss the different ways of assessing and reducing them. The techniques available to do this are :

1. experience based rules
2. scoping calculations
3. comparison with test results
4. model improvement/ hierarchical modelling
5. sensitivity analysis.

Certain techniques can be applied before discretisation and meshing such as experience and scoping calculations. However, techniques such as sensitivity studies and model improvements require an F.E. result set.

The choice of treatment used is very much governed by the error source and the physical model being analysed. Table 1.1 shows which error treatment technique is applicable to each source.

Error Source	Control Technique				
	1	2	3	4	5
a) Mathematical Model	×	×	×	×	NA
b) Domain	×	(×)	NA	×	×
c) Fluid Properties	×	×	×	NA	×
d) Boundary Conditions	×	×	×	×	×

Table 1.1: Application of Error Control Techniques

2.2.1 Experience Based Rules

Experience manifests itself in the knowledge of the analyst and also from existing test results. It can be applied to treat all of the 4 idealisation error sources. It is the most relevant technique with regards to controlling errors. The only drawback of using experience is that it itself is not controllable. This is due to the very fact that it 1) relies on making a judgement on how relevant it is to the situation being analysed and 2) whether it is sufficiently accurate information. However, it is good practise to use experience at an early stage before time is committed to discretisation and meshing .

2.2.2 Scoping Calculations

These calculations are based on simple conceptual models of fluid behaviour and are used to back up engineering judgement and screen out significant effects. They are useful in deciding what would be an appropriate model to use for the analysis. However, scoping isn't very appropriate for controlling domain errors. For example in deciding what effect simplifying the geometry would have on errors, scoping calculations are not detailed enough to provide such information. In fact, feature modelling within the FE analysis itself can only do that.

2.2.3 Comparison with Test Results

The advantage of comparing FE results with test results is that it allows a direct and independent estimate of error. But the liabilities are that tests introduce their own errors, can be very expensive and are often not practicable.

Nevertheless, if test results are available they can be used to regulate errors originating from the assumptions made on a mathematical model, material properties and boundary conditions.

2.2.4 Hierarchical Modelling

This procedure is a progressive process which involves solving a problem at one level of idealisation and then resolving it (or a part of it) on a more detailed representation. By comparing the results the analyst should be able to assess how to improve the first solution.

2.2.5 Sensitivity Analysis

This technique doesn't give a measure of the idealisation error directly. Its application, however, will indicate the significance of certain parameters on the result and so highlight how to improve the process. It works by making a small change on an input parameter and calculating its effect on a resultant output parameter. One of its usefulness is in the control of errors from assumptions on fluid properties and boundary conditions.

2.3 Application of the Techniques

In this section modelling the flow in a gas turbine combustion chamber will be used as an example to highlight how errors are introduced and how they can be assessed and reduced.

2.3.1 The Mathematical Model

Error treatment on the mathematical model is needed to qualify the behavioural assumptions made. Choosing which mathematical model to use will rely on either experience or test results.

An experienced combustion engineer will have some idea on what model would be appropriate for a particular combustion chamber. If, perhaps it is a new design it could be prudent to carry out a water flow test to determine how the fluid is behaving.

Once the analysis has been run experience is the quickest technique that can be used to assess the validity of the results. As a first pass it is very useful. However, if a higher level of accuracy is needed scoping calculations, test results, or even hierarchical modelling will be needed.

If the results are to be compared with test results then a more sophisticated test than a water analogy test may well have to be performed.

2.3.2 The Domain

The techniques employed to assess errors arising from domain simplification will be reliant on the nature of the simplification. Typical domain assumptions made in combustion can modelling are:

1. geometrical simplification of the burner/ injector unit
2. ending the domain at the combustor exit and making the flow pass into an outflow plenum.

Experience can be used to indicate the extent to which geometrical simplifications can be taken, without gross errors being introduced. But if, for instance, the true nature of the spray cone of the injected fuel needs to be determined then hierarchical modelling may be necessary.

Because of the turbine being upstream of the combustion can it forces the analyst to end the domain at the combustor exit. Consequently the predicted conditions at this plane are not reliable. Experience could possibly be used to assess these errors but test results will reveal more. A hot rake test will indicate the temperature and pressure variations in the exit plane.

2.3.3 Fluid Properties

It will be difficult and very unlikely to know exact values of density and viscosity since they are reliant on knowing the conditions of the flow i.e. the pressure, temperature and velocity. The inlet conditions for the combustor may be determined from simple calculations and/ or test results.

Simple calculations on the upstream compressor can reveal the pressure, temperature and flow rate and as a first pass this will be moderately adequate. At a second pass the analyst could resort to the results from an analysis on the compressor or test results.

Test results will of course have associated experimental errors but if they are carefully accounted for they can be carried forward.

2.3.4 Boundary Conditions

Ordinarily, for a static wall one would apply a no-slip boundary conditions. However with a combustion chamber cooling air is injected through the can walls. Often the assumption is made that this cooling air enters the chamber perpendicularly which will bring errors on how the flow mixes with the hot gases.

To get a good boundary layer prediction a large number of cells will be needed. This, though, is not always possible because of constraints on time and computational costs. This being the case errors are clearly introduced. Judgement can be used to assess these error.

Chapter 3 Fluid Theory

3.1 Fluid Flow Fundamentals

The FLOTRAN fluid element FLUID141 is used for the calculation of two dimensional distribution of velocity and pressure for single phase Newtonian fluid. It also allows thermal effects to be modelled if present. The flow problem is defined by the laws of conservation of mass, momentum and energy, which are expressed in terms of partial differential equations. These equations are then discretised with a finite element based technique.

3.1.1 Continuity Equation

Conservation of mass law for two dimensional flow produces:-

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho V_x)}{\partial x} + \frac{\partial(\rho V_y)}{\partial y} = 0 \quad [3.1]$$

where V_x and V_y = components of velocity vector in the x and y directions

respectively

ρ = density

t = time

The rate of change of density can be replaced by:-

$$\frac{\partial \rho}{\partial t} = \frac{\partial \rho}{\partial P} \frac{\partial P}{\partial t} \quad [3.2]$$

Since, for the compressible algorithm,

$$\rho = \frac{P}{RT} \Rightarrow \frac{\partial \rho}{\partial P} = \frac{1}{RT} \quad [3.3]$$

Where P = pressure
 R = gas constant
 T = temperature

In the case of an incompressible fluid,

$$\frac{d\rho}{dP} = \frac{1}{\beta} \quad [3.4]$$

where β = user defined input quantity; default equal 10^{15} .

This value implies that pressure waves will travel infinitely fast through entire problem domain for a perfectly compressible fluid.

3.1.2 Momentum Equation

The relationship between the stress and the rate of deformation of the fluid is given by:-

$$\tau_{ij} = -P\delta_{ij} + \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \delta_{ij} \lambda \frac{\partial u_i}{\partial x_i} \quad [3.5]$$

where τ_{ij} = stress tensor

u_i = orthogonal velocities ($u_1 = V_x, u_2 = V_y, u_3 = V_z$)

μ = dynamic viscosity

λ = second coefficient of viscosity

The product of the second coefficient of viscosity and the divergence of the velocity is zero for a constant density fluid and considered to be small and therefore neglected in a compressible fluid.

The momentum equation without further assumptions regarding properties, are as follows:-

$$\begin{aligned} \frac{\partial \rho V_x}{\partial t} + \frac{\partial (\rho V_x V_x)}{\partial x} + \frac{\partial (\rho V_y V_x)}{\partial y} = \\ \rho g_x - \frac{\partial P}{\partial x} + R_x + \frac{\partial}{\partial x} \left(\mu_e \frac{\partial V_x}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu_e \frac{\partial V_x}{\partial y} \right) + T_x \end{aligned} \quad [3.5]$$

$$\begin{aligned} \frac{\partial \rho V_y}{\partial t} + \frac{\partial (\rho V_x V_y)}{\partial x} + \frac{\partial (\rho V_y V_y)}{\partial y} = \\ \rho g_y - \frac{\partial P}{\partial y} + R_y + \frac{\partial}{\partial x} \left(\mu_e \frac{\partial V_y}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu_e \frac{\partial V_y}{\partial y} \right) + T_y \end{aligned} \quad [3.6]$$

where

g_x, g_y = components of acceleration due to gravity (body forces)

ρ = density

μ_e = effective viscosity which is equal to dynamic viscosity for laminar flow

R_x, R_y = distributed resistance which any further source terms required

i.e. flow through porous media that is not included in the modelling of the geometry.

T_x, T_y = viscous loss terms.

The viscous loss terms are eliminated in the incompressible constant property case. The order of the differentiation is reversed in each term, reducing the term to a derivative of the continuity equation which is zero.

$$T_x = \frac{\partial}{\partial x} \left(\mu \frac{\partial V_x}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial V_y}{\partial x} \right) \quad [3.7]$$

$$T_x = \frac{\partial}{\partial x} \left(\mu \frac{\partial V_x}{\partial y} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial V_y}{\partial y} \right) \quad [3.8]$$

The conservation of energy can be expressed in terms of the stagnation temperature for highly compressible flow or static temperature for low speed incompressible analyses.

3.1.3 Energy Equation

3.1.3.1 Compressible

The compressible energy equation in terms of stagnation temperature is given by:-

$$\begin{aligned} \frac{\partial}{\partial t} (\rho C_p T_o) + \frac{\partial}{\partial x} (\rho V_x C_p T_o) + \frac{\partial}{\partial y} (\rho V_y C_p T_o) = \\ \frac{\partial}{\partial x} \left(K \frac{\partial T_o}{\partial x} \right) + \frac{\partial}{\partial y} \left(K \frac{\partial T_o}{\partial y} \right) + W^v + E^k + Q_v + \Phi + \frac{\partial P}{\partial t} \end{aligned} \quad [3.9]$$

where C_p = specific heat capacity at constant pressure

T_o = stagnation temperature

K = thermal conductivity

W^v = viscous work

Φ = viscous heat generation term

Q_v = volumetric heat source

E^k = kinetic energy

The static temperature (T_s) is given by:-

$$T_s = T_o - \frac{V^2}{2C_p} \quad [3.10]$$

where V = the magnitude of the fluid velocity

The viscous term is given by:-

$$W^v = V_x \mu \left[\frac{\partial^2 V_x}{\partial x^2} + \frac{\partial V_x}{\partial y^2} + \frac{\partial}{\partial x} \left(\frac{\partial V_x}{\partial x} + \frac{\partial V_y}{\partial y} \right) \right] + V_y \mu \left[\frac{\partial^2 V_y}{\partial x^2} + \frac{\partial V_y}{\partial y^2} + \frac{\partial}{\partial y} \left(\frac{\partial V_x}{\partial x} + \frac{\partial V_y}{\partial y} \right) \right] \quad [3.11]$$

Expressed in tensor notation as :-

$$W^v = u_j \mu \left[\frac{\partial}{\partial x_i} \frac{\partial u_j}{\partial x_i} + \frac{\partial}{\partial x_k} \frac{\partial u_k}{\partial x_j} \right] \quad [3.12]$$

where the repetition of subscripts implies a summation over the three orthogonal directions.

The kinetic energy term is given by :-

$$E^k = -\frac{\partial}{\partial x} \left[\frac{K}{C_p} \frac{\partial}{\partial x} \left(\frac{1}{2} |V^2| \right) \right] - \frac{\partial}{\partial y} \left[\frac{K}{C_p} \frac{\partial}{\partial y} \left(\frac{1}{2} |V^2| \right) \right] \quad [3.13]$$

The viscous dissipation term is given by :-

$$\Phi = 2\mu \left[\left(\frac{\partial V_x}{\partial x} \right)^2 + \left(\frac{\partial V_y}{\partial y} \right)^2 \right] + \mu \left[\left(\frac{\partial V_y}{\partial x} + \frac{\partial V_x}{\partial y} \right) \right] \quad [3.14]$$

Expressed in tensor form as :-

$$\Phi = \mu \left(\frac{\partial u_i}{\partial x_k} + \frac{\partial u_k}{\partial x_i} \right) \frac{\partial u_i}{\partial x_k} \quad [3.15]$$

3.1.3.2 Incompressible

The energy equation for the incompressible equation may be derived by neglecting the viscous work (W^v), the pressure work, viscous dissipation (Φ) and the kinetic energy (E^k) from the compressible case. As the kinetic energy is neglected, the static temperature (T_s) and the stagnation temperature (T_o) are the same. The energy equation then takes the form of a thermal transport equation for the static temperature.

$$\begin{aligned} \frac{\partial}{\partial t}(\rho C_p T_s) + \frac{\partial}{\partial x}(\rho V_x C_p T_s) + \frac{\partial}{\partial y}(\rho V_y C_p T_s) = \\ \frac{\partial}{\partial x} \left(K \frac{\partial T_s}{\partial x} \right) + \frac{\partial}{\partial y} \left(K \frac{\partial T_s}{\partial y} \right) + Q_v \end{aligned} \quad [3.16]$$

3.2 Discretisation of Equations

The momentum, energy, species transport and turbulence equations all have the same form of the scalar transport equation. Within this equation, there are four types of terms;

transient, advection diffusion and source. The descitisation method can be described by referring the variable considered as (ϕ) . The form of the scalar transport equation is :-

$$\begin{aligned} \frac{\partial}{\partial t}(\rho C_{\phi} \phi) + \frac{\partial}{\partial x}(\rho V_x C_{\phi} \phi) + \frac{\partial}{\partial y}(\rho V_y C_{\phi} \phi) + \frac{\partial}{\partial z}(\rho V_z C_{\phi} \phi) = \\ \frac{\partial}{\partial x} \left(\Gamma_{\phi} \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left(\Gamma_{\phi} \frac{\partial \phi}{\partial y} \right) + \frac{\partial}{\partial z} \left(\Gamma_{\phi} \frac{\partial \phi}{\partial z} \right) + S_{\phi} \end{aligned} \quad [3.17]$$

where

C_{ϕ} = transient and advection coefficient

Γ_{ϕ} = diffusion coefficient

S_{ϕ} = source terms

Since the approach is the same for each equation, only the generic transport equation need be treated. Each of the four types of terms will be outlined in turn. The discretisation process therefore consists of deriving the element matrices to put together the matrix equation.

$$\left([A_e^{transient}] + [A_e^{advection}] + [A_e^{diffusion}] \right) \{\phi_e\} = \{S_e^{\phi}\} \quad [3.18]$$

Galerkin's method of weighted residuals is used to form the element integrals. Denote by W^e the weighting function for the element which is also the shape function.

3.2.1 Transient Term

The transient contribution to the matrix is of the form:-

$$[A_e^{transient}] = \int W^e \frac{\partial(\rho C_{\phi} \phi)^e}{\partial t} d(vol) \quad [3.19]$$

A lumped mass approximation is used so that

$$\int W^e \frac{\partial(\rho C_\phi \phi)^e}{\partial t} d(vol) = \frac{\partial(\rho C_\phi \phi)^e}{\partial t} \int W^e d(vol) \quad [3.20]$$

A backward difference scheme is used to evaluate the transient derivative and is expressed as ,

$$\frac{\partial(\rho\phi)}{\partial t} = \frac{(\rho\phi)_{n-2}}{2\Delta t} - \frac{4(\rho\phi)_{n-1}}{2\Delta t} + \frac{3(\rho\phi)_n}{2\Delta t} \quad [3.21]$$

The n th time step produces a contribution to the diagonal of the element matrix, while the derivative from the previous time step form contributions to the source term.

3.2.2 Advection Term

The advection term is handled through a monotone streamline approach based on the idea that pure advection transport is along characteristic lines. It can be viewed in terms of a quantity being transported in a known velocity field. In pure advection transport, one assumes that no transfer occurs across characteristic lines and can be expressed in terms of streamline velocities,

$$\frac{\partial(\rho V_x \phi)}{\partial x} + \frac{\partial(\rho V_y \phi)}{\partial y} + \frac{\partial(\rho V_z \phi)}{\partial z} = \frac{\partial(\rho V_s \phi)}{\partial s} \quad [3.22]$$

when expressed along a streamline, is constant through out an element,

$$[A_e^{advection}] = \frac{d(\rho V_s \phi)}{ds} \int W^e d(vol) \quad [3.23]$$

This formulation is made for every element, each of which will have only one node which gets contributions from inside the element. The derivative is calculated using a simple difference,

$$\frac{d(\rho V_s)}{ds} = \frac{(\rho V_s \phi)_U - (\rho V_s \phi)_D}{\Delta s} \quad [3.24]$$

where

D = subscript for value at the downstream node

U = subscript for value taken at the location at which the streamline through the downwind node enters the element.

Δs = distance from the upstream point to the downstream node

The value of the upstream location is unknown but can be expressed in terms of the unknown nodal values it is between. The process consists of cycling through all the elements and identifying the down wind nodes. A calculation is made based on the velocities to see where the streamline through the downwind node came from. The weighting factors are calculated based on the proximity of the upwind location to the neighbouring nodes.

3.2.3 Diffusion Terms

These terms comes from the integration over the problem domain after the multiplication by the weighting function.

$$\begin{aligned} \text{Diffusion contribution} = \int W^e \frac{\partial}{\partial x} \left(\Gamma_\phi \frac{\partial \phi}{\partial x} \right) d(vol) + \int W^e \frac{\partial}{\partial y} \left(\Gamma_\phi \frac{\partial \phi}{\partial y} \right) d(vol) \\ + \int W^e \frac{\partial}{\partial z} \left(\Gamma_\phi \frac{\partial \phi}{\partial z} \right) d(vol) \end{aligned} \quad [3.25]$$

Each component is treated the same way. For illustration, consider the x component and integrating it by parts;

$$\int W^e \frac{\partial}{\partial x} \left(\Gamma_\phi \frac{\partial \phi}{\partial x} \right) d(vol) = \int \frac{\partial W^e}{\partial x} \Gamma_\phi \frac{\partial \phi}{\partial x} d(vol) \quad [3.26]$$

Once the derivative of ϕ is replaced by the nodal values and the derivatives of the weighting function, the nodal values will be removed from the integrals.

$$\frac{\partial \phi}{\partial x} = W_x^\phi \phi \quad [3.27]$$

$$W_x^e = \frac{\partial W^e}{\partial x} \quad [3.28]$$

$$[A_e^{diffusion}] = \int (W_x^e \Gamma_\phi W_x^e + W_y^e \Gamma_\phi W_y^e + W_z^e \Gamma_\phi W_z^e) d(vol) \quad [3.29]$$

3.2.4 Source Terms

The evaluation of the source terms consists of multiplying the source terms by the weighting function and integrating over the volume.

$$S_\phi^e = \int W^e S_\phi d(vol) \quad [3.30]$$

3.3 Solution Algorithm

ANSYS/FLOTRAN uses a segregated sequential algorithm as a solution technique. This means that the element matrices are formed, assembled and the resulting system is solved for each degree of freedom separately. The development of the matrices proceeds in two parts:-

- A. The form of the equations is achieved and the terms are evaluated.
- B. The segregated solution algorithm is outlined and the element matrices are developed from the equations.

3.3.1 Segregated Solution Algorithm

Each degree of freedom is solved in a sequential manner. The equations are coupled, so that each equation is solved with intermediate values of the other degrees of freedom. The process of solving all the equations in turn and then updating the properties is called the global iteration. The global iterative procedure can be summarised as :-

1. Formulate -solve V_x equation approximately
2. Formulate - solve V_y equation approximately
3. Formulate - solve V_z equation approximately
4. Formulate Pressure equation using velocities
5. Solve pressure equation for pressure.
6. Update velocities based on approximated velocities and pressure
7. Formulate - Solve energy equations for temperature
8. Solve species transport equation
9. Update temperature dependent properties
10. Solve turbulence equations
11. Update effective properties based on turbulence solution
12. Check rate of change of the solution
13. End of global iteration

For a detail description of this technique, refer to reference [2]

Chapter 4 Backward Facing Step

4.1 Model Definition

In order to assess the sensitivity of the finite element method to element distortions/perturbations, it was necessary to model a fluid mechanical problem which has available, either a closed form solution or experimental results. This would enable a model calibration to be made and providing a standard from which deviations can be measured.

The backward facing step has provided this means in the form of experimental results. It has been a standard benchmark widely used to evaluate the capabilities of flow codes to predict separated flows for different types of flow conditions. There exists, numerous experimental data of velocity profiles, stream functions etc., albeit in graphical format. Denham and Patrick[2] are one of these examples; they provide results for steady-state low Reynolds number, laminar flow for an incompressible two dimensional fluid.

4.1.1 Domain Definition :

The analysis is performed on the entire structure without making any assumptions on the limits of the domain i.e. with regard to where the boundaries are to be drawn.

The simplicity of the geometry also makes it quite easy to model using regular shaped elements and therefore avoiding the need for any further geometric simplifications or domain assumptions which may lead to possible errors.

The fluid to be modelled is water for which properties at different temperatures are widely available in most property handbooks [3]. The flow regime is low speed and laminar which is a computationally stable problem and requiring relatively short run times. In laminar flow analyses, the velocity is ordered and smooth, due to its high viscous, slow-moving nature. It is an incompressible problem since density fluctuations are neglected. The temperature equation for this type of flow neglects kinetic energy changes and viscous dissipation, making the mathematical formulation somewhat simpler to solve.

4.1.2 Definition of Boundary Conditions :

Since the geometry represents the complete structure, boundary conditions can be precisely applied. Both walls are static thus no slip boundary can be applied as assumptions related to wall temperatures are not required. Inlet conditions would be difficult to apply since it is based on the experimental as opposed to the theoretical fully developed flow. However, this can be overcome by taking values from the experimental curves provided and interpolating these values onto the analysis model.

Inaccuracies involved in this procedure will be due to the technique used to impose these conditions and not because of the problem type. Refer to section 5.2. This, nonetheless is not a concern since the experimental results are only to be used to calibrate the analysis model and can be eliminated once this has been achieved.

The far field wall condition can also be a possible source of error. Since, if it is not located sufficiently distant from the point of interest, it can cause erratic results to occur. It is therefore important to ensure that it is appropriately located. This was achieved by varying the far fields location until the aerodynamic flow parameters in the region of concern became unchanged.

4.2 Anticipated Results :

The flow is fully developed on arrival at the channel inlet. Separation is expected to occur at the step edge and the separation stream lines should carry straight on parallel to the wall before dipping towards the re-attachment point on the lower wall. With the higher Reynolds numbers, there should be a tendency for separation to occur on the upper wall. As re-development of the velocity profiles begins, the magnitudes of the streamlines should then increase back to the fully developed laminar state.

4.3 Presented Results :

Denham and Patrick's experiment was performed for water at Reynolds numbers of 73, 125, 191 and 229. The results presented, are a set of velocity profile plots at different stations along the axial length of the channel for the various flow speeds. Plots of flow streamlines at Reynolds numbers of 73 and 229 and graphical plots characterising length measurements with numerical predictions at the recalculation zone just downstream of the step.

Chapter 5 Sensitivity Analysis

5.1 Model Domain

The model was created in MSC PATRAN version 6.0. The dimensions are represented in figure 5.1.

It is clear from figure 5.1 that the analysis model's domain is smaller than that of the experiment. The experiment required a longer inlet (200 mm) flow channel to ensure that the flow had stabilised and fully developed before reaching the step. This was not necessary in the case of this analysis and therefore the problem could be rectified by simply applying a fully developed inlet boundary condition. However, it was rather more important for the exit condition to ensure that the boundary was sufficiently far to prevent adverse influence on the region of interest; namely, at the step location.

The origin was located at the base of the step, consistent with the experimental results and the model sub-divided so that results could be extracted at the same experimental locations to enable comparisons to be made.

5.2 PATRAN Model

The sub-division of the domain is illustrated in figure 5.2, where PATRAN surfaces were created for each region. The surfaces were then meshed using the isomesh option, producing regular 2-D linear quadrilateral elements and ensuring that the boundary

between surfaces were maintained. The model was equivalenced; removing duplicate nodes on the surface boundaries and the appropriate element checks were then made. These checks included skew angle, aspect ratio, warp, taper ratio, reversed normals and element connectivity.

The final model contained plain strain 2-D linear quadrilaterals, since PATRAN did not support ANSYS FLOTRAN fluid elements. As a consequence, boundary conditions and appropriate fluid material properties could not be applied. A neutral (ASCII) file and an ANSYS input deck was then written.

The ANSYS input deck was edited to remove the structural solution parameters. This also included changing the element type to a FLUID141 element. The modified input deck was then read into ANSYS and the inlet nodes selected and written out into an output file.

The program "DEVFLW", used to determine the fully developed inlet flow conditions, was then run using the nodal output file as input. DEVFLW (see Appendix A.1.1. for program listing of DEVFLW) contains the measured experimental data of the fully developed inlet flow for Reynolds numbers of 73, 125 and 191 from reference [2] in the form of data statements. It calculates and then interpolates the axial velocities from the measured values onto the nodal positions provided. A file is then written which contains, in the ANSYS command form, the interpolated velocities. This file is then added to the original ANSYS input deck.

The above modified deck was once again read into ANSYS. The remaining boundary conditions with fluid properties and the FLOTRAN solution parameters were applied. In order to achieve optimum running conditions, the deck was first run interactively with the residuals monitored, particularly that of pressure. By adjusting various execution parameters, an optimum convergence rate was achieved.

These execution parameters were copied from the ANSYS log file and added to the modified deck; parameters that were subsequently used for the different runs.

5.3 Element FLUID141

FLUID141 can be used to model transient or steady state fluid/thermal systems that involve fluid and/or non-fluid regions. The conservation equations for viscous fluid flow and energy are solved in the fluid region, while only the energy equation is solved in the non-fluid region. This element is used to solve for flow and temperature distributions within a region, as opposed to elements that model a network of one-dimensional regions hooked together.

The element is defined by three nodes (triangle) or four nodes (quadrilateral) and by isotropic material properties. The co-ordinate system can be selected via a switch on the element definition command, and may be either Cartesian, axisymmetric or polar.

5.3.1 Input Data

If the material number of a FLUID141 element is set to 1, it is assumed to be a fluid element. Its properties, density, viscosity, thermal conductivity and specific heat will need to be defined. Only one fluid can be analysed and must be in a single phase and the thermal conductivity and specific heat are relevant only if the problem is thermal in nature. A turbulence model can be used to determine the kinetic energy and dissipation rate degrees of freedom.

5.3.2 Output Data

The solution output associated with this element, takes the form of nodal quantities. The degrees of freedom are supplemented by intermediate properties and derived quantities. Additional output is provided via a file containing periodic tabulations of the maximum, minimum, and average values of the velocities, pressure, temperature, turbulence quantities and properties. Also included are the convergence monitoring parameters calculated at every global iteration, the mass flow at all the inlets and outlets and the heat transfer information at all boundaries.

A wall results file is also produced containing information related to the boundary faces of wall elements. Average pressure, temperature, shear stress, Y-plus values and wall heat fluxes are stored, along with vectors denoting the normal direction from the surface and the direction of the velocity immediately adjacent to the wall.

5.3.3 Element Assumptions and Restrictions

The element must not have a negative or zero area. The connectivity of an element must be defined with the nodes in anticlockwise order. The element must lie in the X-Y plane. Triangles are treated as degenerated quadrilaterals where the duplicated third node is ignored. Only linear elements are supported. FLUID141 may not be used with any other element.

5.3.4 Element Formulation Assumptions

1. The nodal and global co-ordinate systems must remain the same throughout an analysis.
2. The problem domain and the finite element mesh may not change during an analysis.
3. The fluid is a single phase fluid.

4. Non-fluid elements may not vary with temperature.
5. Free surfaces are not permitted.
6. The equation of state of gases is the ideal gas and is not valid for Mach numbers in excess of 5.
7. In the incompressible option, work done on the fluid by pressure forces, viscous dissipation and kinetic energy terms are neglected in the energy equation. The incompressible energy equation is the thermal transport equation.
8. In the compressible adiabatic case, the total temperature is assumed constant and the static temperature is calculated by subtracting the kinetic term from it.
9. Transient boundary conditions in a transient analyses must be handled with multiple restarts.

For full details refer to reference [4].

5.4 Fluid Properties

The solution option was steady-state, adiabatic, laminar incompressible flow. The fluid was water, with values of density and dynamic viscosity [3] at room temperature equal to 1000m^3 and 0.001 Pas respectively and were kept constant throughout the analysis.

5.5 Boundary Condition

- 5.5.1. A fully developed axial flow velocity (V_x) for $Re = 125$ based on reference [2] was imposed on the inlet nodes; see figure 5.3.
- 5.5.2. Inlet radial velocity (V_y) = 0.0 ms^{-1} .
- 5.5.3. All exterior walls, $V_x = V_y = 0.0\text{ ms}^{-1}$.
- 5.5.4. At exit, gauge pressure (P) = 0.0 Pa ; see figure 5.3.

5.6 Execution Control Parameters

The global iterations of the analysis was set to 3000. The pressure termination criteria of 0.32×10^{-4} based on preliminary convergence assessment run was used, whilst for velocity, the default values were accepted. It was apparent, from the residual monitor, that pressure convergence was the more difficult parameter to satisfy. After several variations in the control parameters, an artificial viscosity of 0.3 eventually provided an optimum convergence rate. For the solver options, the tri-diagonal matrix algorithm (TDMA) was applied to both velocities and the preconditioned conjugate gradient with the maximum number of semi-direct iterations of 1500 was required for the pressure. Refer to Appendix A.2.0. for example of FLOTTRAN input deck.

5.6.1 Tri-diagonal Solver

The method consists of breaking the problem into a series of tri-diagonal problems where any entries outside the tri-diagonal portion of the matrix are treated as source terms using the previous values. The method reduces to Gauss-Seidel iterative process when the mesh is unstructured or for an arbitrarily numbered system.

Since this is not a precise method, the tri-diagonal matrix is not executed to convergence. Instead, the user specifies the number of tri-diagonal sweeps per global iteration. This method is described in detail in Patanker [5].

5.6.2 Preconditioned Conjugate Matrix Solver

The conjugate gradient method is an iterative method, but unlike the tri-diagonal solver, it attempts to calculate an exact solution to the problem's equation set. It is preconditioned with an incomplete Choleski decomposition and is used only for the

pressure equation in incompressible flows. The sequential algorithm must allow space for a non-symmetric coefficient matrix for the momentum and energy equations. For the symmetric matrix, only half this space is required whilst the other half is used to store the decomposition. Computer memory requirement is increased when preconditioning is requested.

See references [6], [7] and [8] for more information.

5.7 Derived Output Quantities

FLOTRAN uses primitive values i.e. axial velocity (V_x), tangential velocity (V_y) and pressure (P) in the solution to the Navier-Stokes equations. From these values, the so called derived quantities can be calculated once convergence has been achieved. The derived quantities requested were stagnation pressure (P_t), stream function, pressure coefficient, Y-Plus and the wall shear stress.

5.8 Model Validation

The boundary conditions and the execution control parameters were set as explained in sections 5.5. and 5.6., the models of the three listed Reynolds numbers ($Re = 73, 125, 191$) were submitted to ANSYS as background jobs.

Once convergence or a successful termination had been established, an ANSYS database was created from the modified input deck. The results file was then read, the 'general-post' option invoked and the results for the last iteration selected. Since the region of largest velocity gradient was in the location of the step i.e. at $x = 12$ mm and $x = 30$ mm, the predicted results were inspected along radial lines at the above mentioned axial positions.

Nodes at these positions were selected and transferred into a nodal results file. The axial velocity for both were printed for ascending values of radial position (y) and written into separate files. The program "INTRP" (see Appendix A.1.3. for program listing of INTRP), which interpolates the experimental results for a given axial position onto the nodes provided was then executed. The predicted results from ANSYS was then appended onto the interpolated results file provided by "INTRP". The awk script "script.awk" (see Appendix A.1.4. for program listing of script.awk), was then used to format the newly appended file to be read by the plotting program "xmgr".

From these preliminary results, the Reynolds number of 125 was decided upon since it presented sufficiently large gradients at the region of interest and was a slightly greater challenge for the code than the lower value. At the Reynolds number of 191, the laminar flow may not have been maintaining its two dimensional effect.

5.8.1 Mesh Independency

In order to ensure that the best results were obtained for the analysis model, it was first necessary to establish mesh independency, i.e. the point at which further mesh refinements did not produce significant improvement in the computed results. To achieve mesh independency, three models were created.

The mesh refinement process is illustrated in figure 5.4.

5.8.1.1 Model Mesh (1)

Model mesh (1) contained 671 elements and 765 nodes, a rather coarse mesh.

It required 761.313 seconds of CPU time to complete 5000 iterations, but did not converge. Figure 5.5 shows a plot of this mesh.

5.8.1.2 Model Mesh (2)

Model mesh (1) contained 2684 elements and 2871 nodes, a medium refined mesh. This was created by dividing each element in model mesh (1) into four parts and is thus shown in figure 5.6.

It required 1201.456 seconds of CPU time to converge after 1517 iterations.

5.8.1.3 Model Mesh (3)

Model mesh (1) contained 10736 elements and 11109 nodes, a fully refined mesh. This was created by dividing each element in model mesh (2) into four parts and is shown in figure 5.7.

It required 27290.225 seconds of CPU time to converge after 4462 iterations. Figure 5.8 and 5.9 show plots of the velocity profiles at $x = 12$ mm and 30 mm respectively.

Comparing both figures with the experimental results, it can be seen that at the region where the axial velocities are in the range of ± 0.002 ; i.e. the step location and between 0.01 to 0.012 there is a marked deviation. This suggests that localised refinement or grid adaptation may improve these results, more so for figure 5.8. where the flow is heavily disturbed due to its proximity to the step. Refinement was not made to the mesh since this was not the aim of the exercise.

In comparing the results for the three mesh densities, there is a noticeable improvement between mesh (1) and (2), whilst between mesh (2) and (3) the change is very small.

Despite the negligible change, mesh (3) required 23 times more CPU time achieve convergence. This demonstrated that mesh independency had been established and hence the density of mesh (2) was accepted.

ANSYS contour plots of velocity and pressure variations are depicted in figures 10 and 11 respectively. For the velocity plot, the minimum value in figure 10 represents the reattachment point. From the velocity vectors plots; figure 12 and 13 the influence of the step is observed to cause the flow to re-circulate. The re-circulating eddy where both components of velocity are zero, can be clearly seen; figure 13.

Figures 14 and 15 show contour plots of the stream function variations, averaged and non-averaged respectively. It shows a smooth distribution for both plots.

5.9 Quadrilateral Element Aspect Ratio

The aspect ratio test is derived from a test proposed by Robinson and Haggemacher [9]. This test is based on a projection plane created by first bisecting the four element edges, creating a point on the plane at the vector average of the corners. The x-axis extends from the point to the bisector on edge 2. The ratio is calculated as the ratio of

the length from the origin to the bisector of edge 2 and the length from the origin to the bisector of edge 3. If the ratio is less than one, then it is inverted.

For normalisation, the aspect ratio is inverted such that it becomes less than or equal to one. This inverted aspect ratio is subtracted from one yielding the normalised aspect ratio. Thus a square element will have a normalised aspect ratio of zero. See figure 5.16.

5.9.1 Methodology For Aspect Ratio Variation

The neutral file for model mesh (2) was used to recreate a PATRAN database. Using the node modify option within PATRAN, the nodes were moved at the boundary of the element at $x = 0$ and the element to the right of it such that its nodes overlapped those of the third element along an axial line, consequently zeroing the surface area of the second.

By repeating this for each element up to the radial limit of the domain, the first aspect ratio of 2.4. for the model was obtained.

Elements of zero surface area were deleted with their corresponding nodes and duplicate nodes between the new bounding elements were removed by using the PATRAN equivalencing option. A neutral file of this modified mesh and an ANSYS input deck were written into output files.

The procedure was then repeated for elements along the axial length up to $x = 30$ mm, the second inspection point. By this stage, files of aspect ratios of 2.4, 3.6, 4.8, 6.0, 7.2, 8.4, 9.6, 10.8 and 12.0:1 were created. With these files edited and modified as per section 5.2., batch jobs were submitted to ANSYS.

Figure 5.17. and 5.18. shows meshes for aspect ratios of 2.4 and 12.0 respectively.

5.9.2 Aspect Ratio Results

Figure 5.19. represents the aspect ratio variation up to the first inspection point; at $x = 12$ mm. The maximum aspect ratio for this region is 4.8:1 which is within the ANSYS acceptable limit of 5:1. Despite being within this limit, there appears to be a gradual degradation of the results with increasing aspect ratio. Figure 5.20. shows a plot magnifying the region of maximum deviation.

At the channel centre line, there is no appreciable difference with a disparity of 0.1 % for aspect ratio of 4.8:1. At the channel outer wall, the change is too small to be of concern. The largest deviations are occurring between the step height and the lower boundary wall, yielding a maximum difference of 74 % at a radial position of 12.5 mm for the same aspect ratio.

The curves fall to the right of mesh_2 curve and become gradually more vertical as the aspect ratio is increased. By increasing the aspect ratio, there is an inevitable reduction in the overall number of elements in the model. In this case, from 2668 to 2636 for ratios of

2.4:1 and 4.8:1 respectively. This reduction, is localised and restricted to the axial direction, i.e. from $x = 0$ to $x = 30$ mm whilst the radial concentration is maintained.

Figure 5.21 represents the aspect ratio variation up to the second inspection point; at $x = 30$ mm. The maximum aspect ratio for this region is 12:1 which is outside the suggested limit. From figure 5.21., it is shown that there is a relatively small deviation of 6.8 % at channel outer wall at a radius of 42 mm, but larger than that at the first inspection point.

At the channel centre line, the disparity is greater still, with a difference of 2.8 % for the largest aspect ratio, shown in figure 5.22. This appears to be dominated by the codes inability to predict the behaviour of the flow in the region of the step where a maximum disagreement of 90.5 % at a radial position of 10 mm is observed to occur. Figure 5.23 shows a magnified plot of the variation of aspect ratio for this region.

In order to obtain the 12:1 aspect ratio, the number of elements were reduced further still to 2540, leading to a severe localised reduction and a substantial rise in the element aspect ratio gradient

Table 5.1. summarises the maximum percentage difference for the various aspect ratios at inspection points one and two.

Aspect Ratio	Mesh		Maximum % Difference			
	No of elements	No of nodes	@ x = 12 mm	Radial Position/m	@ x = 30 mm	Radial Position/m
2.4	2668	2854	22.9	0.01	24.0	0.01
3.6	2652	2837	41.1	0.0125	12.6	0.01
4.8	2636	2820	74.1	0.0125	19.9	0.075
6.0	2620	2803	-	-	22.6	0.005
7.2	2604	2786	-	-	32.7	0.005
8.4	2588	2769	-	-	46.8	0.01
9.6	2572	2752	-	-	68.5	0.01
12.0	2540	2758	-	-	90.6	0.01

Table 5.1 : Deviation from Model; Mesh (2) for Varying Element Aspect Ratio

Contour plots describing the at aspect ratios of 3.6:1, 7.2:1 and 12:1 are shown in figures 5.24-5.41. As the aspect ratio is increased, the reattachment point is observed to move further downstream; figure 5.24, 5.30 and 5.36. The centre of the re-circulating eddy has also drifted downstream as shown in figures 5.26~5.27, 5.32~5.33 and 5.38~5.39. For the averaged and non-averaged stream function variations, figures 5.28~5.29, 5.34~5.35 and 5.40~5.41 the smooth variation previously observed for the unaltered mesh (2) is maintained. The predictions of the maximum and minimum values and their location are similar except for the aspect ratio of 7.2:1 where shift upstream is observed. A closer inspection may be required for this case.

5.9.3 Element Aspect Ratio Gradient

Based on experience with structured grids for finite difference and finite volume computational fluid codes, it is a well established fact that large aspect ratio gradient as opposed to large aspect ratio grids/elements have been responsible for spurious and inaccurate results. For this reason, it was important to eliminate this effect from the effects that were being observed and thus it required the performance of an aspect ratio

gradient investigation. For this investigation, a new model was created, see figure 5.42., which allowed aspect ratios to be varied linearly. A value of 16:1 was used since could be varied linearly and would produce a significant percentage change from the control mesh.

Three meshes were produced with variations into and away from inspection point two. The steep gradient mesh, illustrated by figure 5.43, contained a range of elements in the radial direction with aspect ratios of 16:1 between $x = 0$ and 30 mm.

The medium gradient mesh, illustrated by figure 5.44, was a variation of aspect ratios of 16:1 and 4:1 in both upstream and downstream of inspection point two.

The smooth gradient mesh contained elements with varying aspect ratios of 16:1, 8:1, 4:1 and 2:1 also upstream and downstream. Figure 5.45, illustrates this mesh.

The results from these analyses are shown in figure 5.46, with figure 5.47 focusing on the area of largest disparity. From the above figures, it is shown that as the gradient is reduced the accuracy of the results is improved, albeit by very little, and the curves are observed to approach that representing the control mesh. However, it can be deduced conclusively that the results are primarily dominated by the large aspect ratio element and not the aspect ratio gradient.

5.10 Quadrilateral Element Skew

The skew test is based on a reference frame created by first bisecting the four element edges, creating an origin at the vector average of the corners where the x-axis extends from the origin to the bisector on edge 2. The z-axis is in the direction of the cross product of the x-axis and the vector from the origin to the bisector of edge 3. The y-axis is in the direction of the cross product of the x and z-axis as shown in figure 5.48.

From reference [9], the skew test uses the angle α (a) between the edge 2 and 4 bisector and the test y-axis. The resulting angle is then subtracted from 90 degrees to yield the skew angle.

For normalisation, the skew angle is divided by 90 degrees to yield the skew factor. Thus a square element will have a skew factor of zero. See figure 5.49.

5.10.1 Methodology For Element Skew Variation

The neutral file for model mesh (2) was used to recreate a PATRAN database. Using the node modify option within PATRAN, the nodes were moved and altered to produce skewed elements.

To illustrate the method used, consider a set of eight elements as shown in figure 5.50; step (1). The nodes at the interface between elements E2 and E7 are moved to the right by one element such that the nodes are superimposed onto element E3's interface with E6; step (2), thus producing degenerate quadrilaterals for both E3 and E6.

By moving the left hand upper corner node of E5 and the lower corner node of E4 as indicated by step (3), it causes elements E3 and E6 to collapse i.e. zeroing their surface area.

Repeating this for each element up to the radial limit of the domain, the first mesh of skew angles of 50.2 degrees for the model is obtained.

Elements E3 and E6 were deleted with their corresponding nodes and the duplicate nodes between the new bounding elements were removed by using the PATRAN equivalencing option. This operation reduced the mesh density to six from ten. A neutral file of this modified mesh and an ANSYS input deck were written into output files.

This method was then repeated for elements along the axial length up to $x = 30$ mm, the second inspection point. Files containing models with element skew angles of 50.2, 67.4, 74.5, 78.2, 80.5 and 84.1 degrees were created. These files were then modified as per section 4.2 and batch jobs were submitted to ANSYS. Figure 5.51 and 5.52 show meshes for skew angles of 50.2 and 84.1 degrees respectively.

5.10.2. Element Skew Results

Figure 5.53 represents the skew angle variation from the step to the first inspection point one.

The maximum skew angle here is 67.4 degrees which is outside the ANSYS acceptable limit of 30 degrees. At the channel outer wall and centre line, the difference is negligible. The faceted appearance of the higher skew angle curve is due to a reduced number of elements, a consequence of the operation and hence a reduced number of radial inspection points.

The largest deviations are observed to occur between the step height and the lower boundary wall, yielding a maximum difference of 56.3 % at a radial position of 12.5 mm for the higher skew angle.

Figure 5.54 shows a magnified plot of this region. The curves fall to the right of mesh_2 curve, but to a much lesser degree than that of aspect ratio and still maintaining the curvature of the control curve. The deviation is observed to grow with increasing magnitude of skew angle.

By increasing the skew angle, there is an inevitable reduction in the overall number of elements in the model. In this case, from 2668 to 2652 for angles of 50.2 and 67.4 degrees respectively. This reduction, is localised and restricted to the axial direction, whilst the radial density is unchanged.

The skew angle variation up to inspection point two is shown in figure 5.55. The maximum angle for this region is 84.1 degrees which is outside the ANSYS suggested limit.

At the channel outer wall, the curves are well behaved and are very much superimposed. Deviations begin to occur as the flow approaches the channel centre line where a disparity of 2.8 % at a radial position of 30 mm for the largest skew angle is observed; see figure 5.56.

The magnitude of the discrepancies are greatest at the step location where due to the skew variations, the mesh density is reduced to 2556 element for 84.1 degrees skew angle. A maximum disagreement of 82.9 % at a radial position of 2.50 mm is observed to occur. Figure 5.57 shows a magnified plot of the variation of skew angle for this location. Table 5.2. summarises the maximum percentage difference for the various skew angles at inspection points one and two.

Element Skew	Mesh		Maximum % Difference			
	No of elements	No of nodes	@ x = 12 mm	Radial Position	@ x = 30 mm	Radial Position
40.2	2668	2854	18.2	0.075	17.6	0.01
67.4	2652	2837	56.3	0.0125	8.1	0.0075
74.5	2636	2820	-	-	10.97	0.01
78.2	2620	2803	-	-	23.8	0.01
80.5	2604	2786	-	-	44.1	0.01
84.1	2556	2735	-	-	82.9	0.0025

Table 5.2: Deviation from Model Mesh (2) for Varying Element Skew

Contour plots showing variations of the flow parameters such as velocity, pressure and stream function for skew angles 67.4, 78.2 and 84.1 degrees are shown in figures 5.58-5.75. From figures 5.58, 5.64 and 5.70 the magnitudes of the minimum and maximum

velocities are comparable but the axial location is gradually moving downstream with increase in skew angle. This is also observed to be the case for the re-circulating eddy region from the vector plots of figures 5.60~5.61, 5.66~5.67 and 5.72~5.73.

A comparison between the averaged and non-averaged stream function plots; figures 5.62~5.63, 5.68~5.69 and 5.74~5.75 show smooth variations with the magnitude of the maxima and minima are also very similar.

5.11 Quadrilateral Element Taper Ratio

The taper test, also proposed by reference [9], creates four triangles bounded by the element edge and the edges created by connecting the element verification reference origin with the two nodes at the element edge. The resulting four triangular areas are calculated and then summed.

The ratio of the area of the smallest triangle and the total area of the element is taken as the taper ratio.

For normalisation, the taper ratio is subtracted from one to yield the taper factor. Thus a square element has a taper factor of zero. See figure 5.76.

5.11.1 Methodology For Taper Ratio Variation

The neutral file for model mesh (2) was used to recreate a PATRAN database. Using the node modify option within PATRAN, the nodes were moved and altered to produce tapered elements.

To illustrate the method used, consider a set of ten elements as shown in figure 5.77; step (1). The nodes at the interface between elements E1 and E10 were displaced to the right by a distance equal to the length of element E2 with the node at the interface between

elements E5 and E6 being displaced to the left a distance equal to the E4 element; illustrated by step (2). This produced degenerate quadrilaterals for E2, E4, E7 and E9.

The nodes at the lower corners and upper corners of elements E3 and E8 respectively were moved in directions indicated by step (3), thus collapsing elements for E2, E4, E7 and E9. By repeating this for each element up to the radial limit of the domain, the first mesh of containing elements with a taper ratio of 0.5 were obtained.

These collapsed elements were deleted with their corresponding nodes and the mesh equivalenced. A neutral file of this modified mesh and an ANSYS input deck were created.

This method was then repeated for elements along the axial length up to the second inspection point. Files containing models with element taper ratios of 0.5, 0.33, 0.25 and 0.2 and 0.5 were created. These files were then modified as per section 5.2. and batch jobs were submitted to ANSYS.

Figure 5.78 and 5.79 show meshes for taper ratios of 0.5 and 0.2 respectively.

It is obvious that by distorting an element, its bounding elements due to interconnection will be distorted also. With this in mind it is important that modifications effected, produced a mesh that comprised predominantly of elements possessing the desired characteristics.

5.11.2 Taper Results

Figure 5.80 represents the taper ratio variation from the step to the first inspection point one. The taper ratio here is 0.5 which is outside the ANSYS acceptable limit of 0.8.

At the channel outer wall and centre line, the difference is negligible. The faceted appearance of the curve is due to a reduced number of elements as a consequence of the operation and hence a reduced number of radial inspection points.

The largest deviations are observed to occur between the step height and the lower boundary wall, yielding a maximum difference of 45.8 % at a radial position of 10.0 mm. Figure 5.81 shows a magnified plot of this region.

The taper curve falls to the right of the mesh_2 curve, but to a much lesser degree than that of both aspect and taper ratios but still maintaining the curvature of the control curve. The deviation is observed to grow with increasing magnitude of taper.

By increasing the taper ratio, there is an inevitable reduction in the overall number of elements in the model. The number of elements are reduced from 2652 to 2620 when the taper is increased from 0.5 to 0.3 respectively. This reduction, is localised and restricted to the axial direction only.

The variation of taper ratio up to the second inspection point is shown in figure 5.82

The maximum taper for this region is 0.2 which is outside the recommended ANSYS limit. At the channel outer wall and centre line, the curves are well behaved and very much superimposed except for the taper ratio of 0.2, see figure 5.83 For this value, the curves appear faceted due to the fewer radial inspection points. Deviations begin to occur as the flow approaches the step location.

The magnitude of the discrepancies become apparent at the step where, due to the taper variations, the mesh density is reduced to 2556 element for 0.2 taper ratio. A maximum disagreement of 53.4 % at a radial position of 10.0 mm is observed to occur.

Figure 5.84 shows a magnified plot of the variation of taper ratio for this location.

Table 5.3. summarises the maximum percentage difference for the various taper ratios at inspection points one and two.

Element Taper	Mesh		Maximum % Difference			
	No of elements	No of nodes	@ x = 12 mm	Radial Position	@ x = 30 mm	Radial Position
0.5	2652	2837	45.8	0.01	13.7	0.01
0.3	2620	2803	-	-	14.1	0.0125
0.25	2588	2769	-	-	31.1	0.01
0.2	2556	2735	-	-	53.4	0.01

Table 5.3 : Deviations from Model Mesh (2) for Varying Element Taper

Variations of flow parameters such as pressure, velocity and stream function for taper ratios of 0.3 and 0.2 are shown as contour plots in figures 85-96. For figures 85 and 91, illustrating the velocity contour distribution, there is a drift of the maximum values location downstream whilst their magnitudes are similar.

The eddy re-circulation represented by figures 5.875.88 and 5.93~5.94 appear erratic. This is due to the reduced number of elements in this region, however the centre remains visible.

For the stream function plots of figures 91~92 and 95~97, the averaged and non-averaged plots show a relatively smooth variation.

5.12 Perturbed Element Downstream of Inspection Point

As a matter of completeness, it was important to ascertain, whether a perturbed element downstream of the inspection point would influence the results upstream. For this experiment, the condition which produced the largest deviations was used. This was the aspect ratio variations.

Elements possessing aspect ratios of 12:1 were created as per section 5.9.1. immediately downstream of inspection point two. The model was modified according to section 5.2.1. and submitted to ANSYS. The results at inspection point two were obtained and plotted.

Figure 5.97 shows the axial velocity profile for this case; at $x = 30.0$ mm. The two curves overlay exactly at the channel outer wall and centre line. A small deviation is observed to occur at step location, yielding a difference of 11.4 % at a radial position of 2.5 mm. Figure 5.98 shows a magnified plot of the axial velocity profile at the step region.

Conclusions

The principal conclusions drawn, can be summarised as follows :-

Aspect Ratio :

1. There is a deterioration in the results with increasing aspect ratio
2. The deterioration is observed to grow as the poor aspect ratio element's proximity draws closer to the inspection point. .
3. The error is greatly affected by this type of distortion.
4. The errors are dominated by the element distortion but, the reduction in the number of elements has also an affect which is difficult to quantify.

Aspect Ratio Gradient :

1. The effect on the error due to the steep gradient is relatively small
2. The inaccuracy is predominantly controlled by the poor aspect ratio elements.

Skew Angle :

1. The error increases as the skew angle is increased.
2. The deterioration is proportional to the proximity of the poor skew angle element to the inspection point.
3. The observed inaccuracies are restricted to the region of the geometry where the flow is highly disturbed.

Taper Ratio :

1. The error is directly proportional to the taper ratio of the element.
2. The error is relatively less affected by this type of distortion than the two previous.

Downstream Perturbations :

1. Element perturbations downstream of the inspection point has a negligible affect on the flow solution.

Despite the results becoming less accurate with increasing distortion, convergence was achieved in every case.

Further Work

The original intention for this thesis was to produce a FORTRAN code where by the above investigations can be performed in a visible manner i.e. access to modifications can be easily made. However, for many and different reasons this could not be achieved and hence a commercial code (ANSYS/FLOTRAN) was used.

For future work I would recommend that a code be produced which enable the following work to be carried out.

1. Assess whether higher flow speeds affect element distortion errors.
2. The effect of using isoparametric elements.
3. The effect of altering the mid-side node position of these isoparametric elements.
4. Altering the direction of element distortion i.e. across flow field as opposed to along flow direction.
5. Mesh adaption.

REFERENCES

1. SAFESA Technical Manual
2. Denham and M. A. Patrick'
"Laminar Flow Over a Downstream-Facing Step in a Two Dimensional Flow Channel", Trans. Instn Chem. Engrs, Vol. 52, 1974 pp. 361-367
3. Robert H. Perry and Don Green,
"Perry's Chemical Engineer's Handbook",
Mc Graw-Hill International Edition, 1984.
4. ANSYS CFD FLOTTRAN Users Guide. SAS IP, Inc.
5. S. V. Patankar, "Numerical Heat Transfer and Fluid Flow", Hemisphere, New York, 1980.
6. Magnus. R. Hestenes and Eduard Stiefel, "Methods of Conjugate Gradients for Solving Linear Systems", Journal of Research of the National Bureau of Standards, Vol. 49, No. 6, 1952.
7. J. K. Reid, "On the Method of Conjugate Gradients for the Solution of Large Sparse Sets of Linear Equations", Proceedings on the Conference on Large Sparse Sets of Linear Equations, Academic Press, 1971, pp. 231-254.
8. H. C. Elman, "Preconditioned Conjugate Gradient Methods for Nonsymmetric Systems of Linear Equations", Advances in Computer Methods for Partial Differential Equations IV, R. Vichnevetsky, E. Stepleman, IMACS, 1981, pp. 409-413.
9. J. Robinson and G. W. Haggemacher, "Element Warning Diagnostics", Finite Element News, June and August, 1982.
10. J. Rea and J. Sykes,
"An Introduction to the Use of the Finite Element Method in Flow Modelling"
AERE R.8322, June 1976
11. C. Taylor, J. O. Medwell, W. D. Morris
"Finite Element Modelling of Flow and Heat Transfer in Turbine Blade Cooling"
Proceedings of the Institution of Mechanical Engineers, C423/037 1991-93
12. R. H. Gallagher, J. T. Oden, C. Taylor, O. C. Zienkiewicz,

"Finite Elements in Fluids -Volume 2", Mathematical Foundations, Aerodynamics and Lubrication, John Wiley and Sons, 1975.

13. Graham. F. Carey, J. Tinsley Oden,

"Finite Elements: Fluid Mechanics Volume VI", Prentice Hall, 1986.

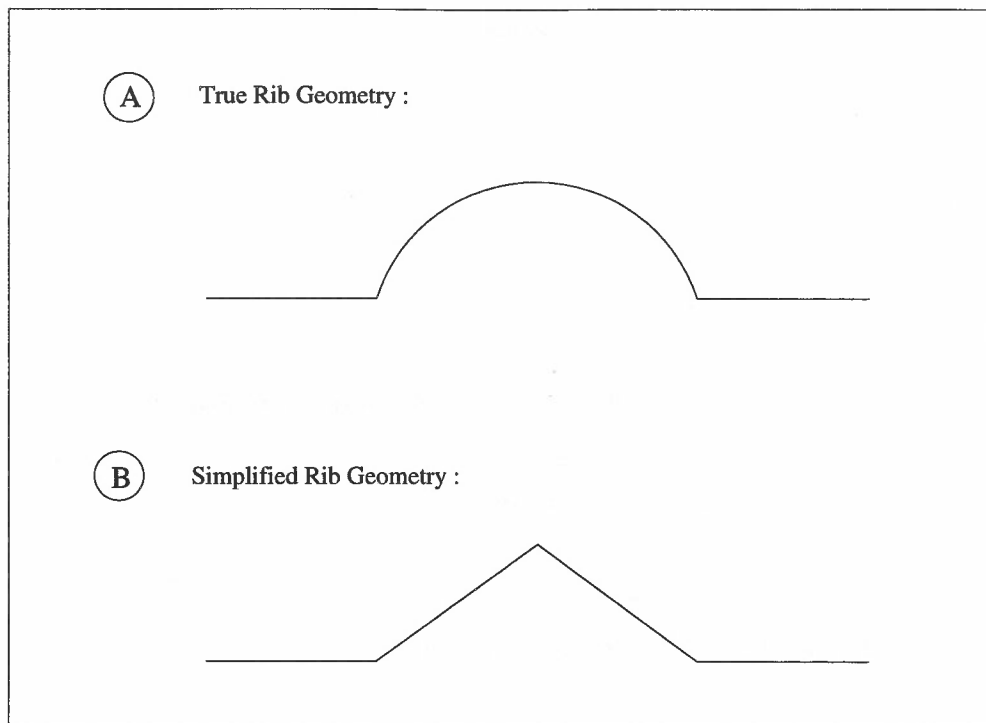


Figure 1.1: Rib Geometry Simplification

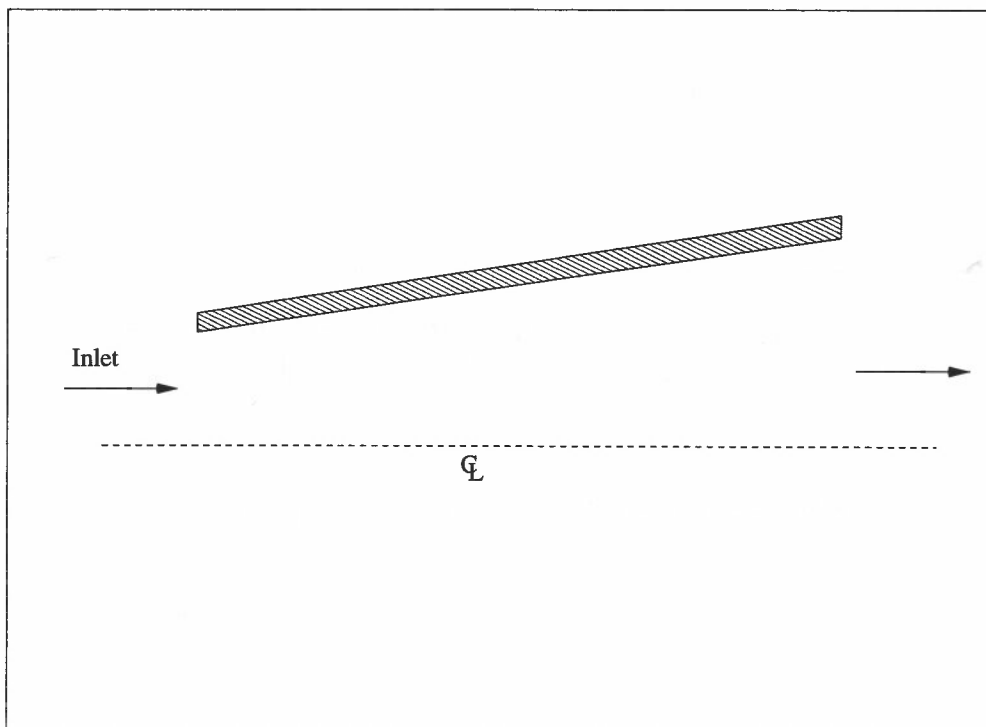


Figure 1.2: Diffuser Boundary Symmetry

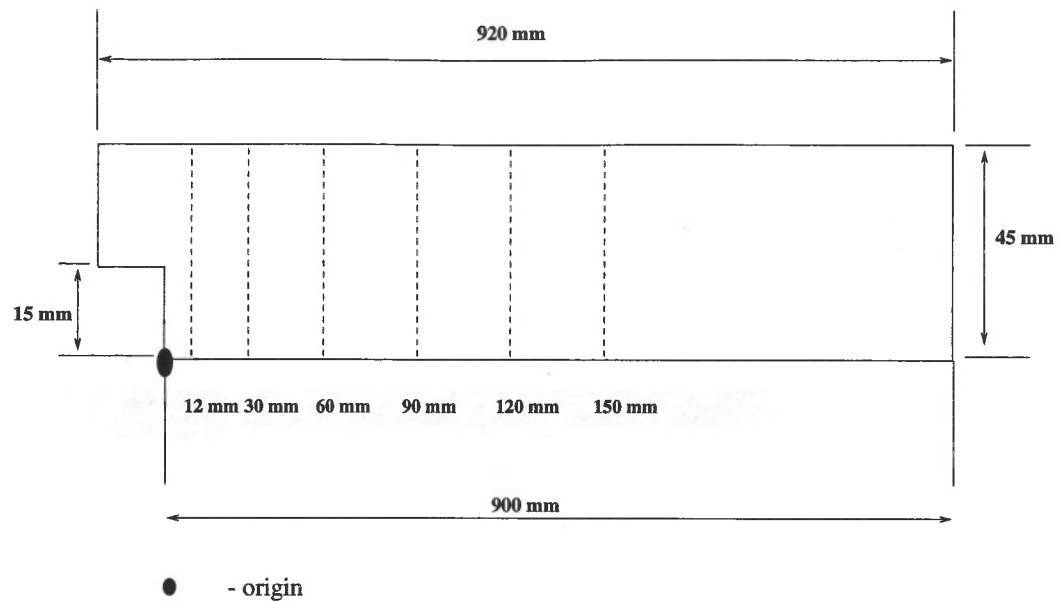


Figure 5.1: Dimensions of Analysis Model Domain

A1	A2	A3	A3	A5	A6	A7	A8
	A9	A10	A11	A12	A13	A14	A15

Figure 5.2: Definition of Surfaces within Patran

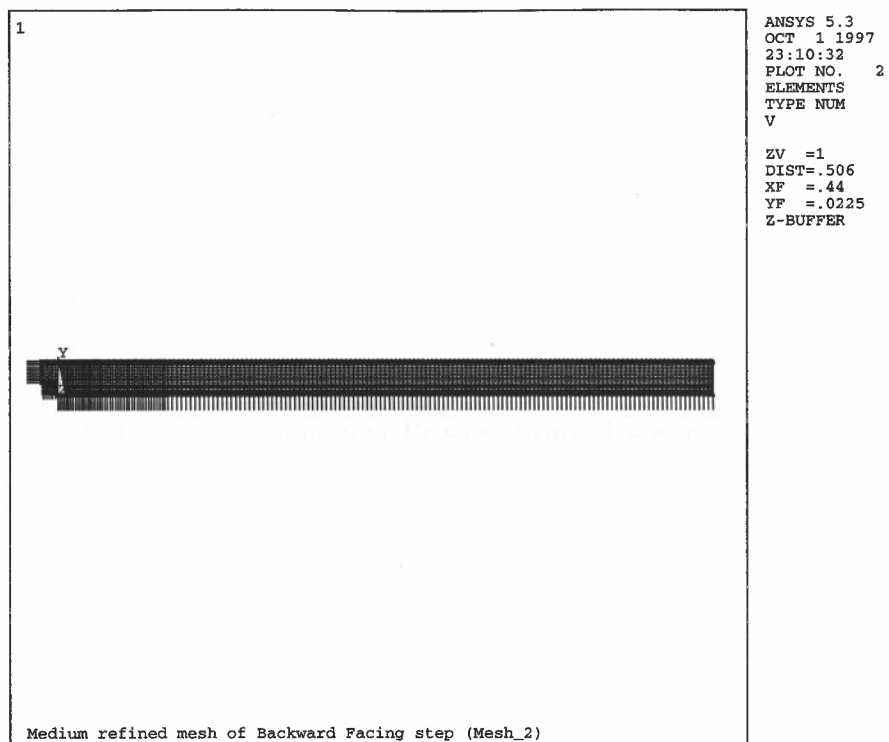
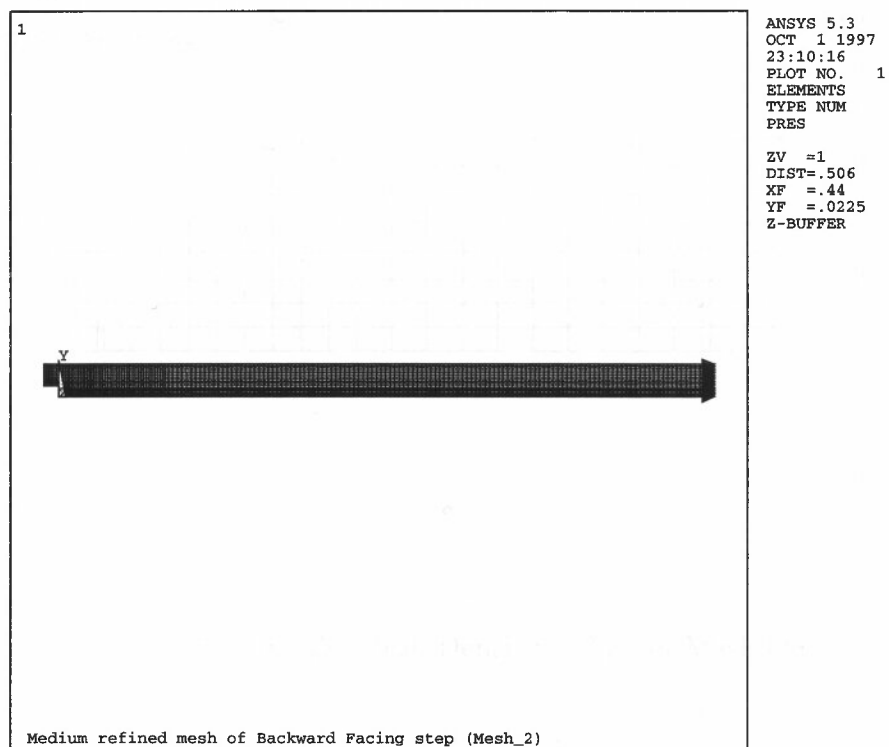


Figure 5.3: Model Boundary Conditions; Velocity and Pressure

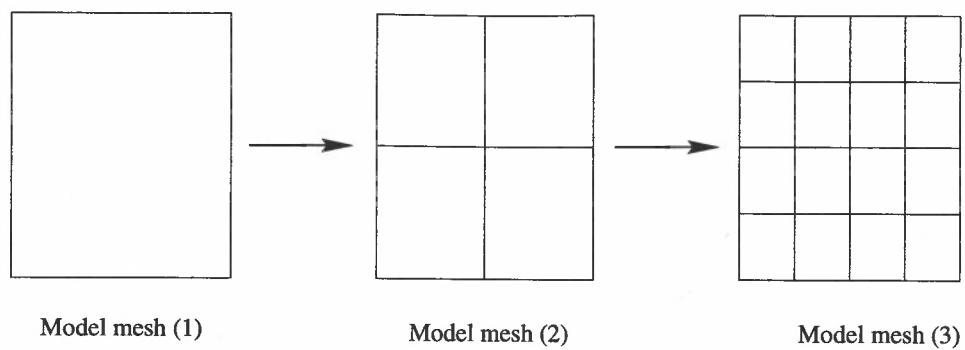


Figure 5.4: Element Refinement Process from Model mesh(1) to Model mesh(3)

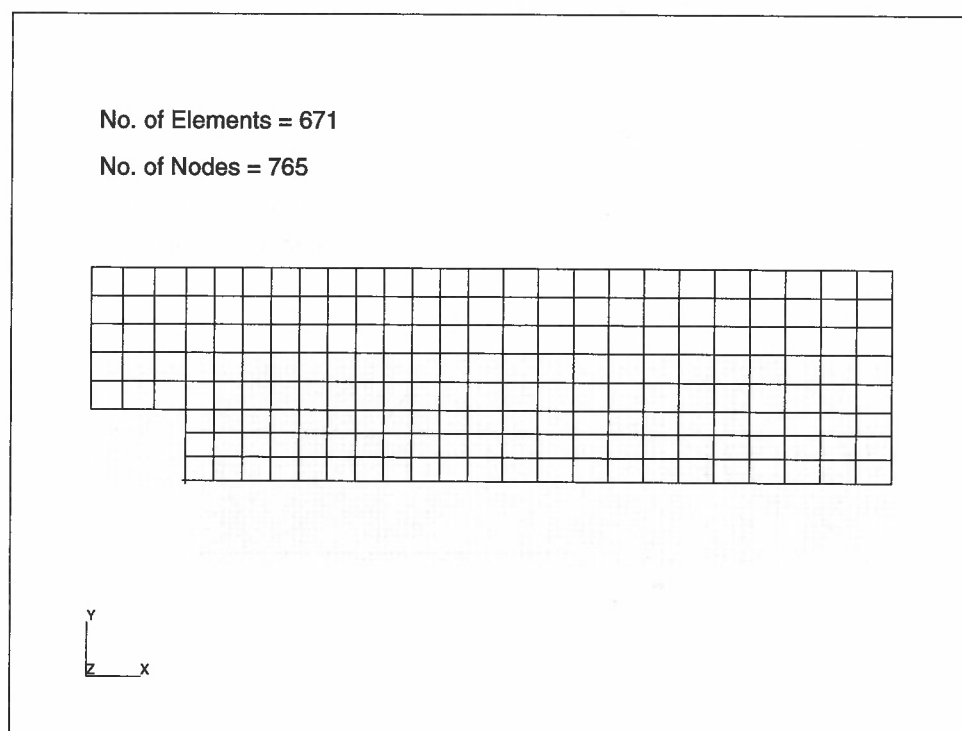


Figure 5.5: Mesh Density of Model Mesh (1)

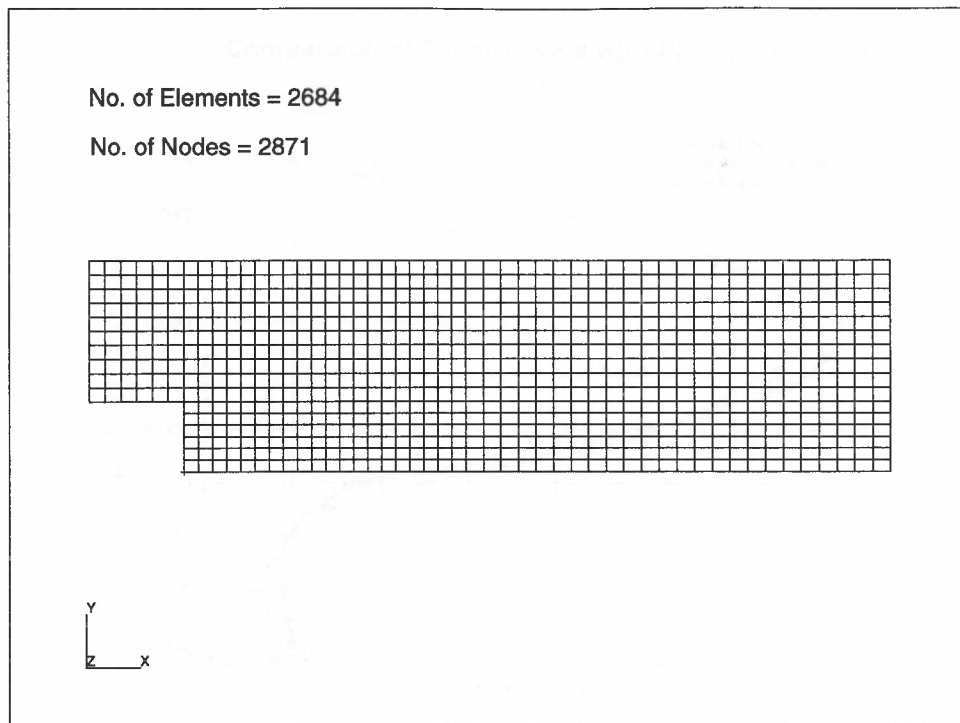


Figure 5.6: Mesh Density of Model Mesh (2)

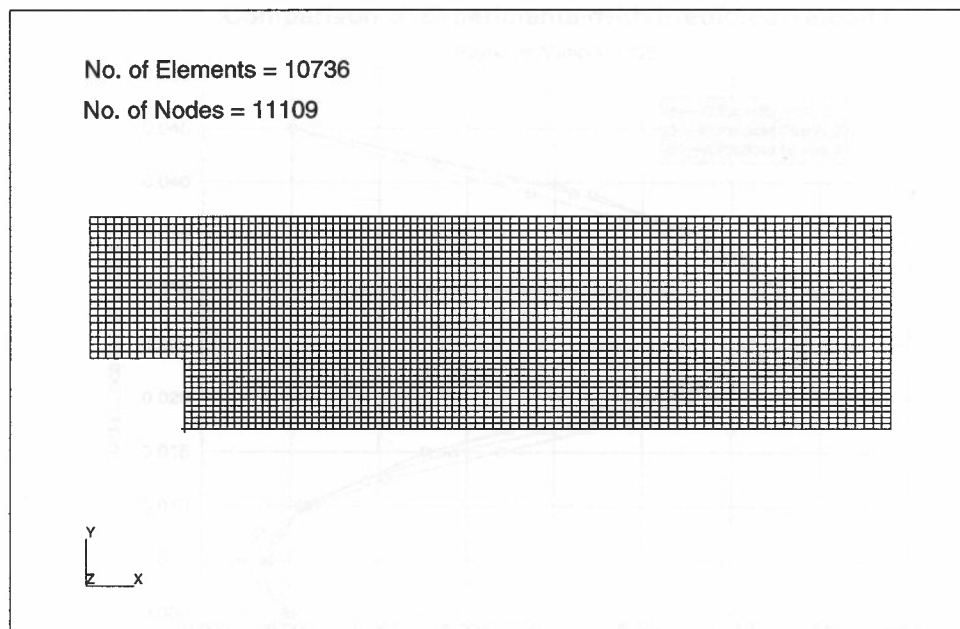


Figure 5.7: Mesh Density of Model Mesh (3)

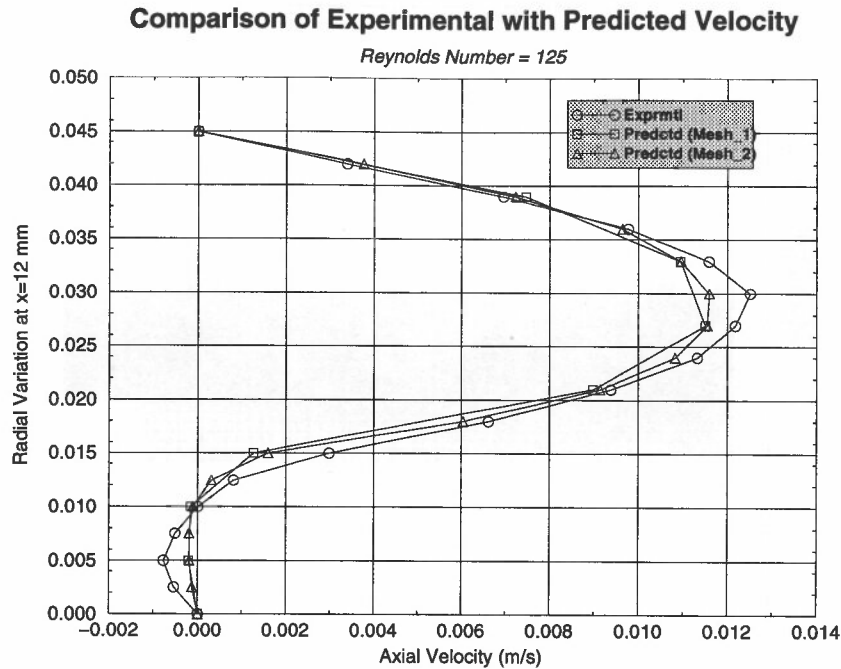


Figure 5.8: Velocity Profile At Three Mesh Densities at $x = 12\text{mm}$

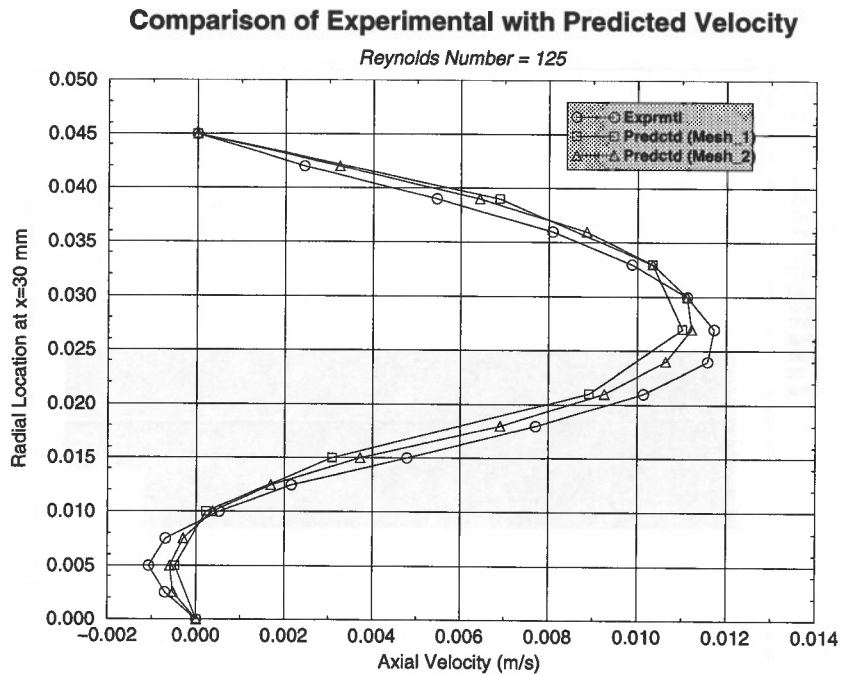


Figure 5.9: Velocity Profile At Three Mesh Densities at $x = 30\text{mm}$

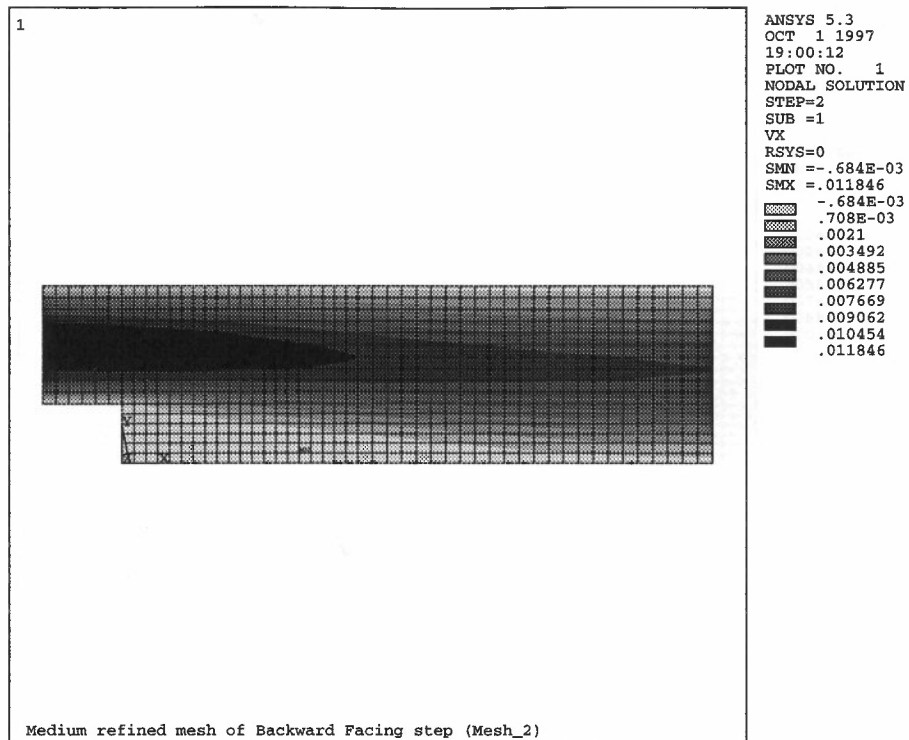


Figure 5.10: Contour Plot of Axial Velocity Variation

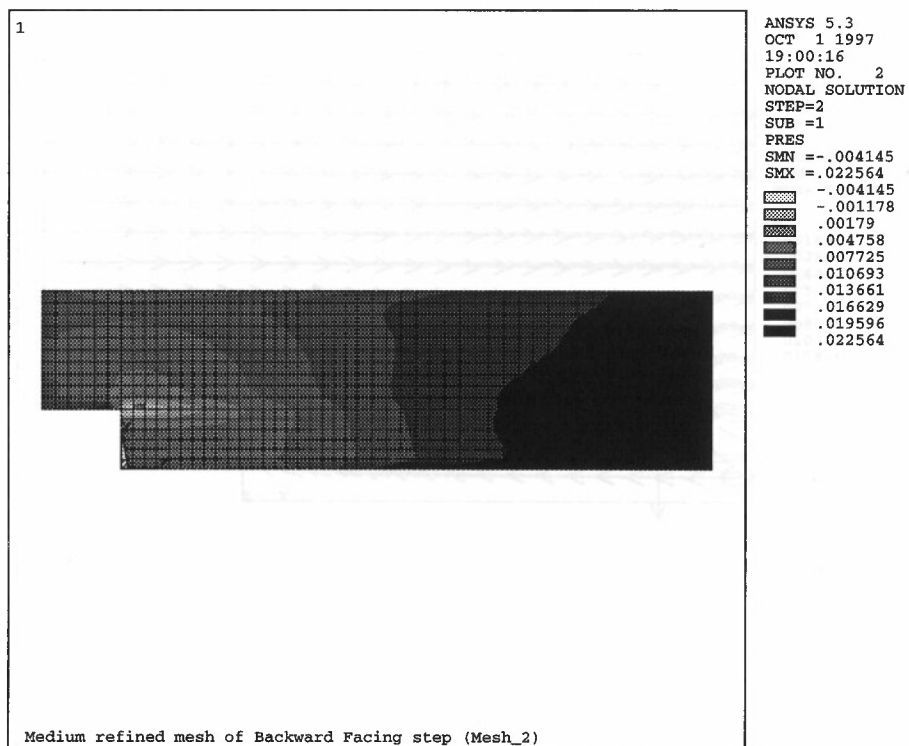


Figure 5.11: Contour Plot of Pressure Variation

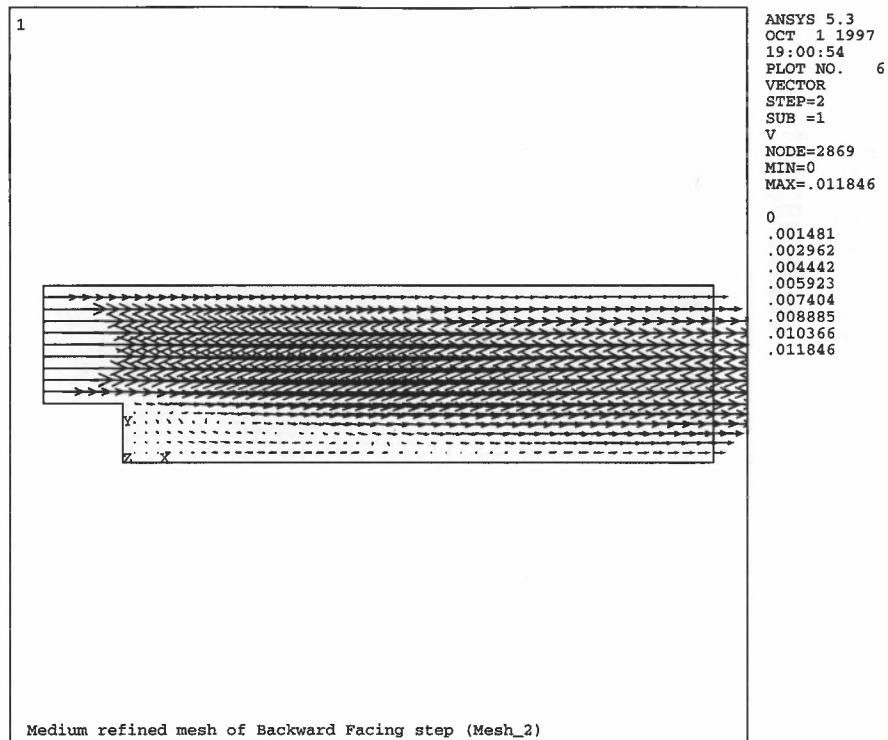


Figure 5.12: Vector Plot of Velocity Variation

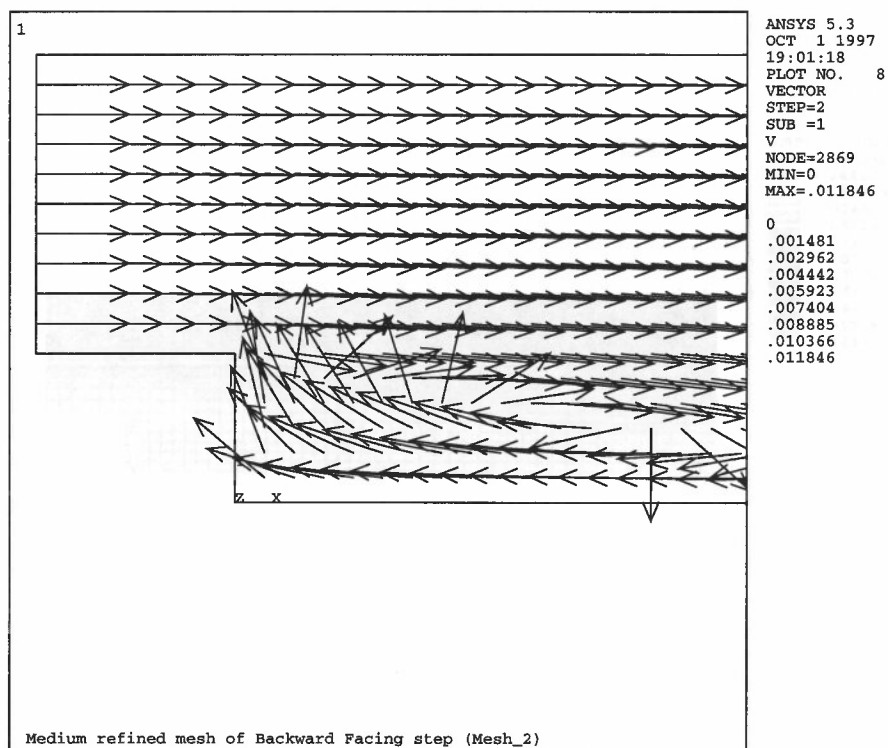


Figure 5.13: Magnification at Step location

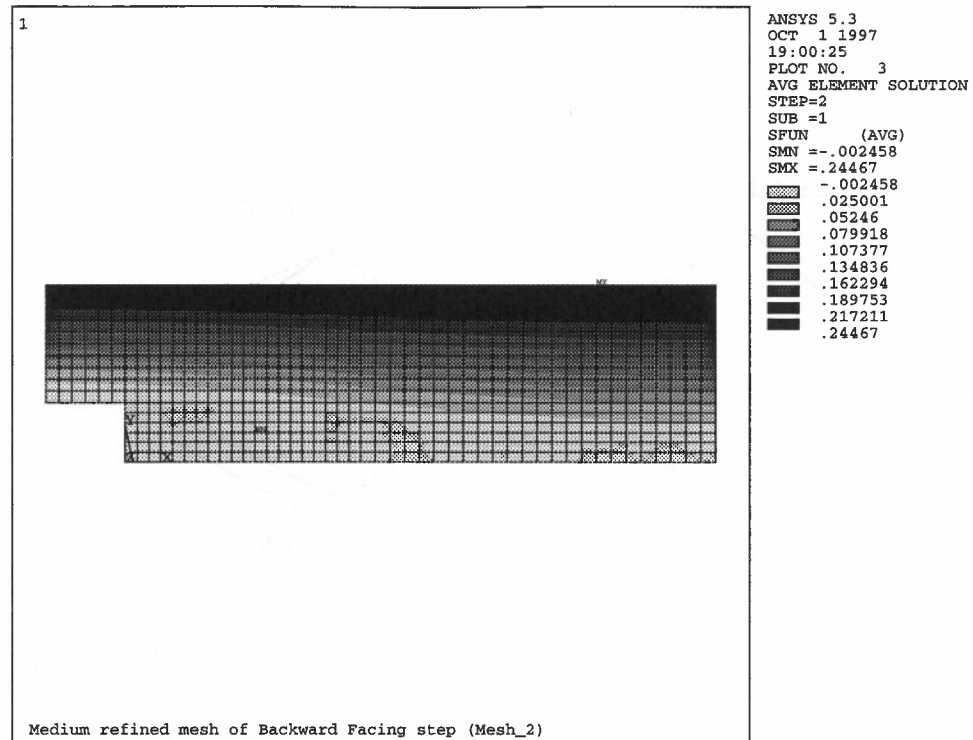


Figure 5.14: Contour Plot of Stream Function Variation; Averaged

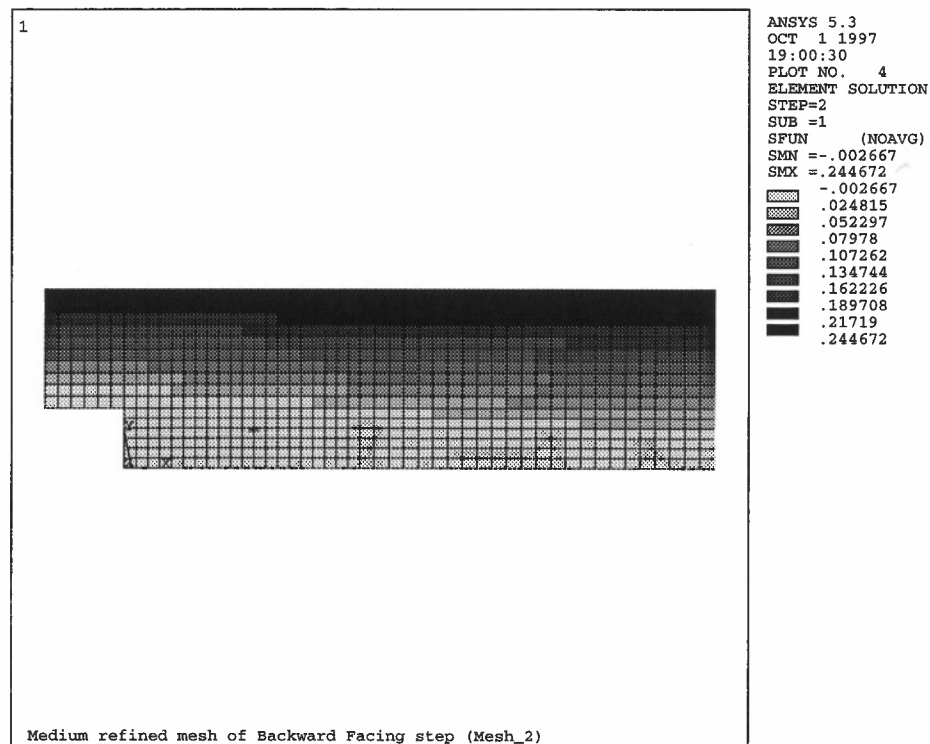


Figure 5.15: Contour Plot of Stream Function Variation; Non Averaged

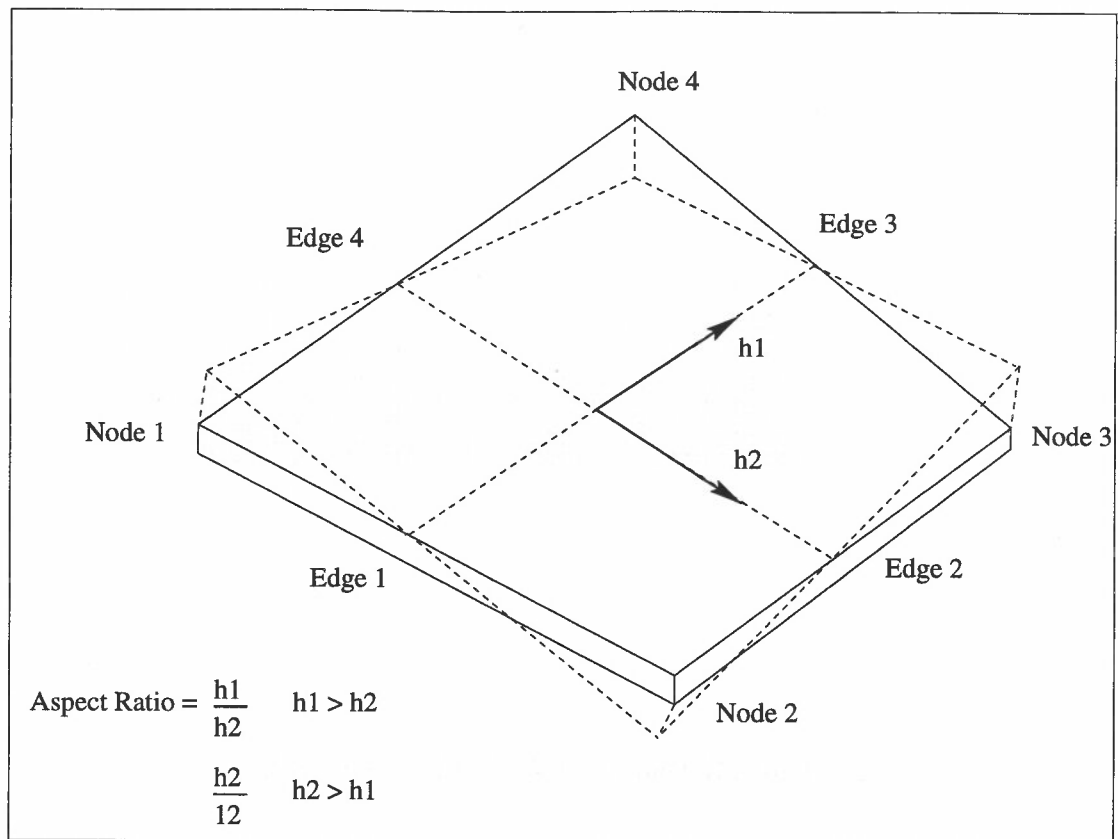


Figure 5.16: Quad Aspect Ratio Definition

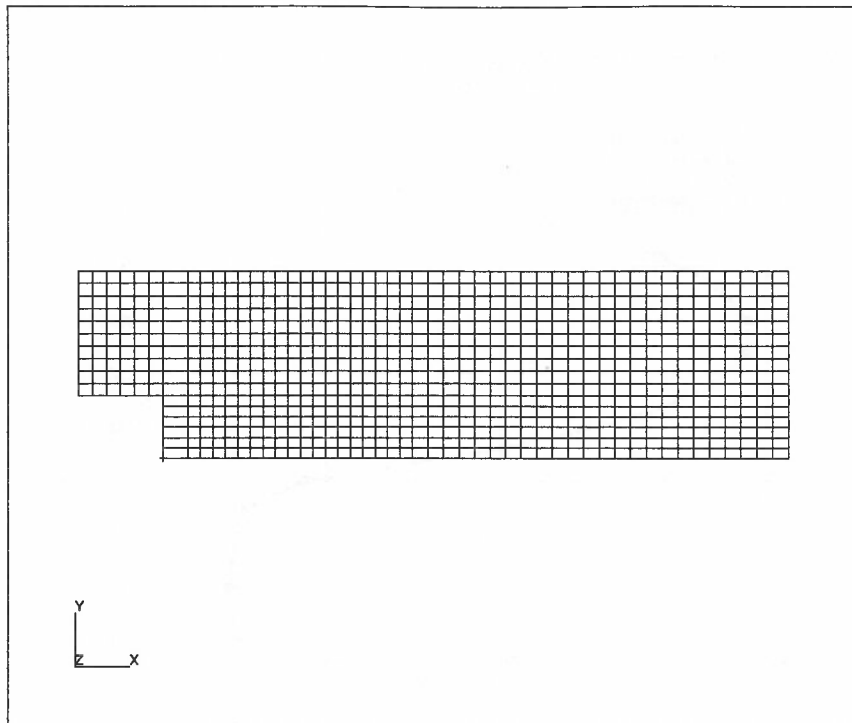


Figure 5.17: Mesh With Aspect Ratio=2.4:1

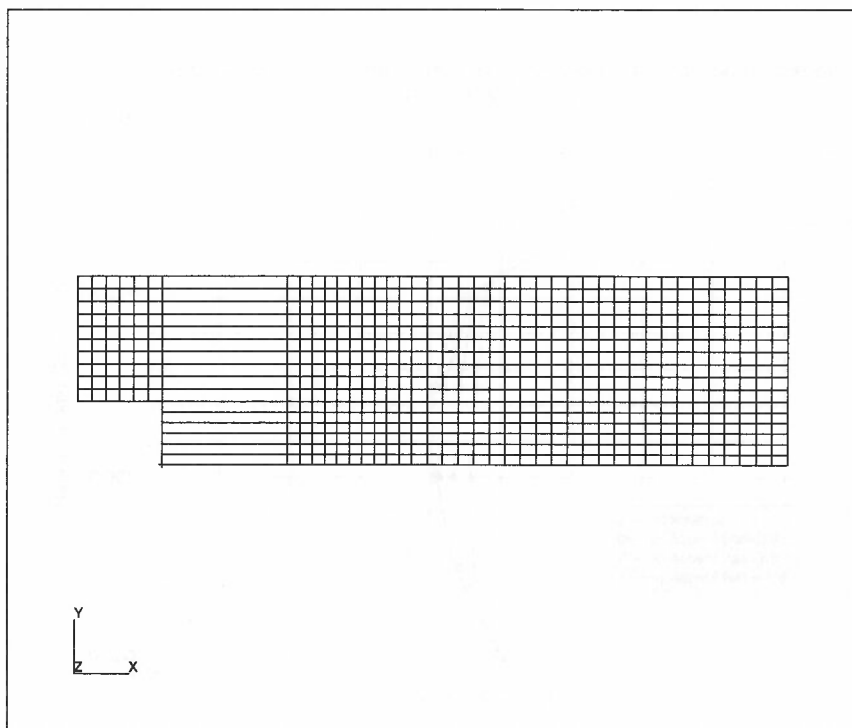


Figure 5.18: Mesh With Aspect Ratio=12:1

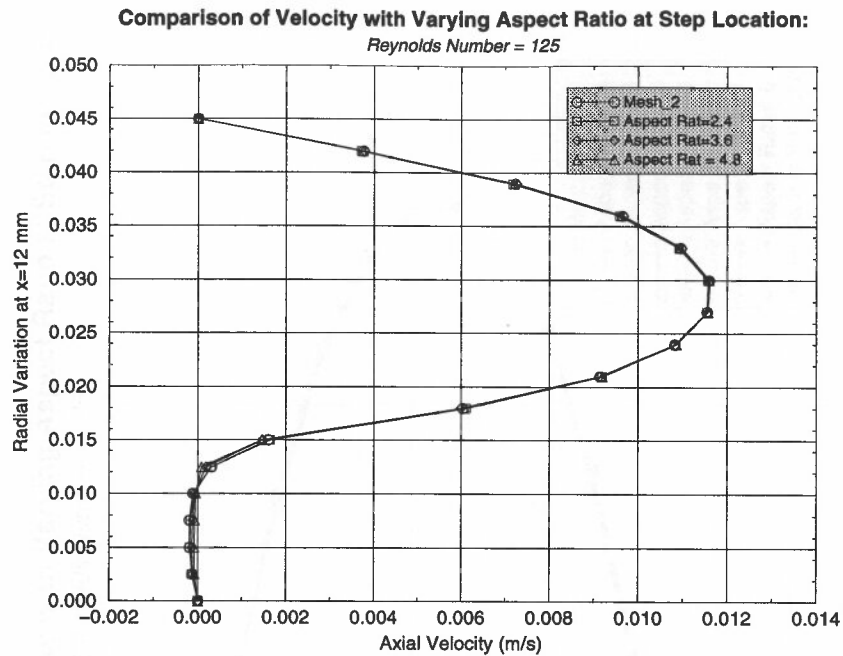


Figure 5.19: Aspect Ratio Variation to Inspection Point One

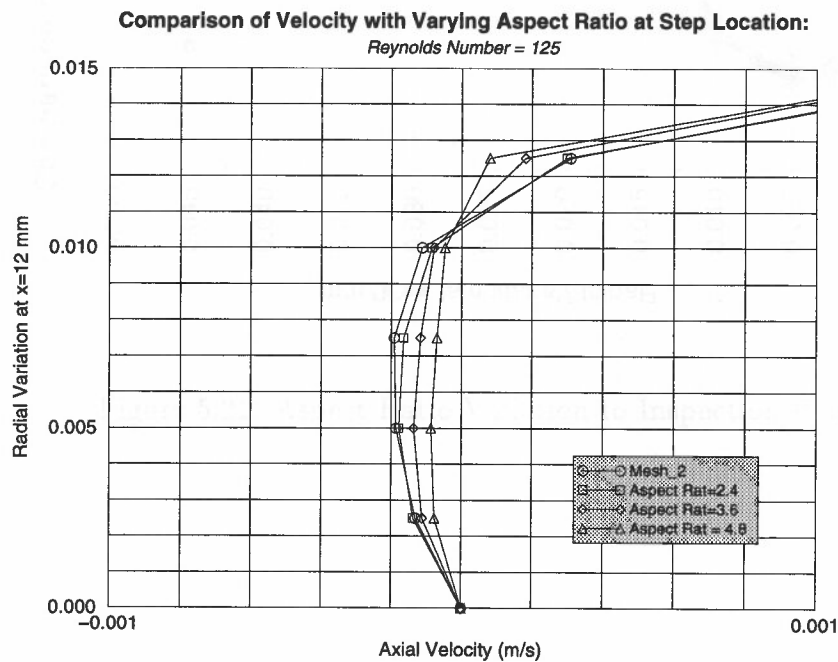


Figure 5.20: Magnification Plot at Step Location

Comparison of Axial Velocity with Varying Aspect Ratio at Step Location

Reynolds Number = 125

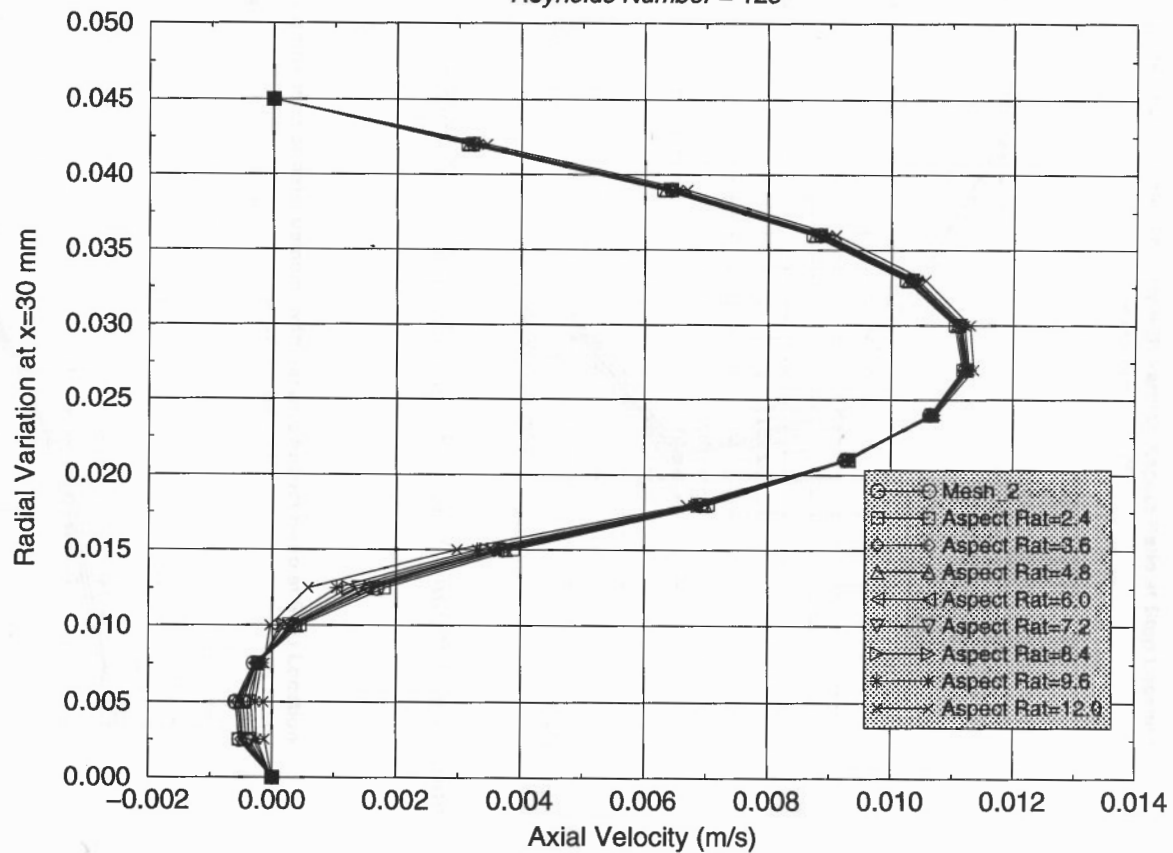


Figure 5.21: Aspect Ratio Variation to Inspection Point Two

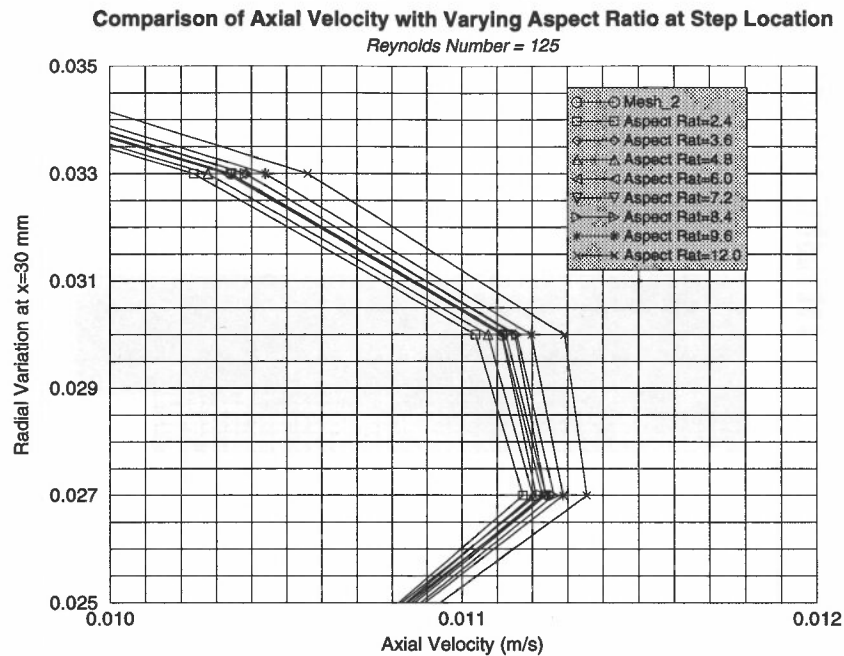


Figure 5.22: Magnification Plot at Channel Centre-line

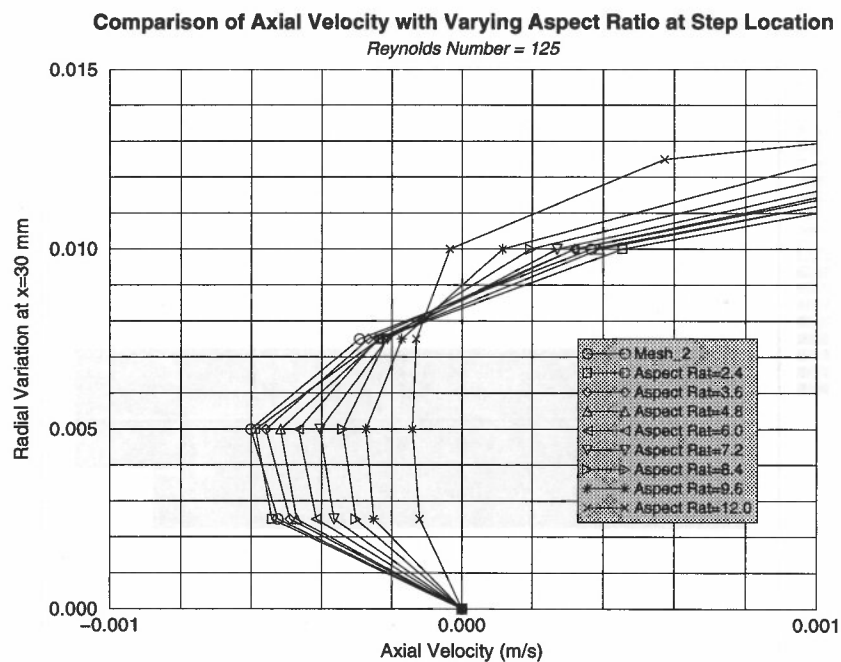


Figure 5.23: Magnification Plot at Step Location

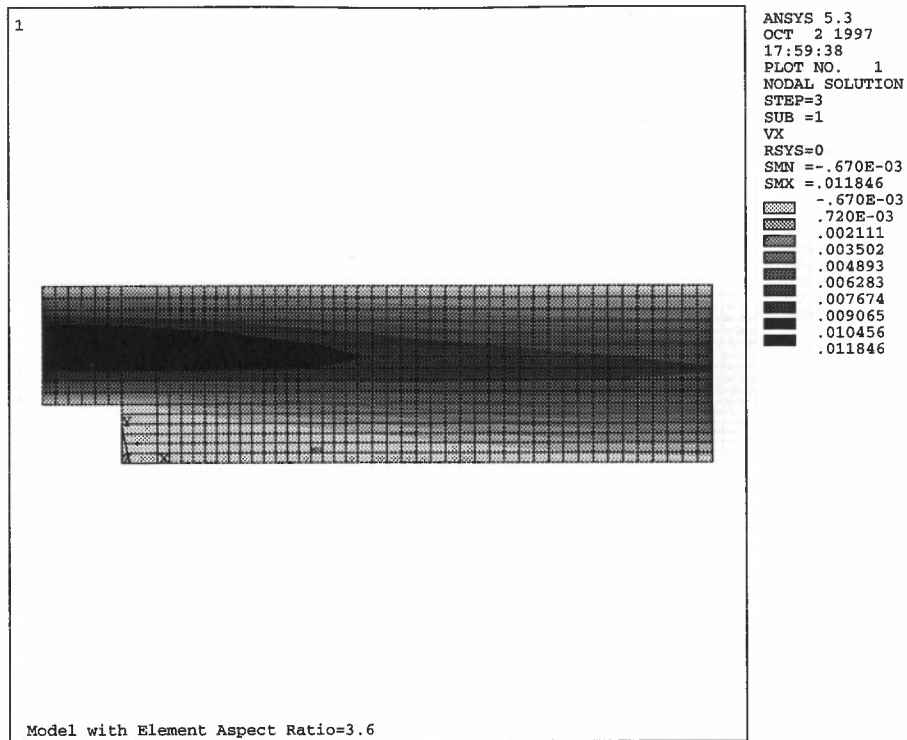


Figure 5.24: Contour Plot of Axial Velocity Variation

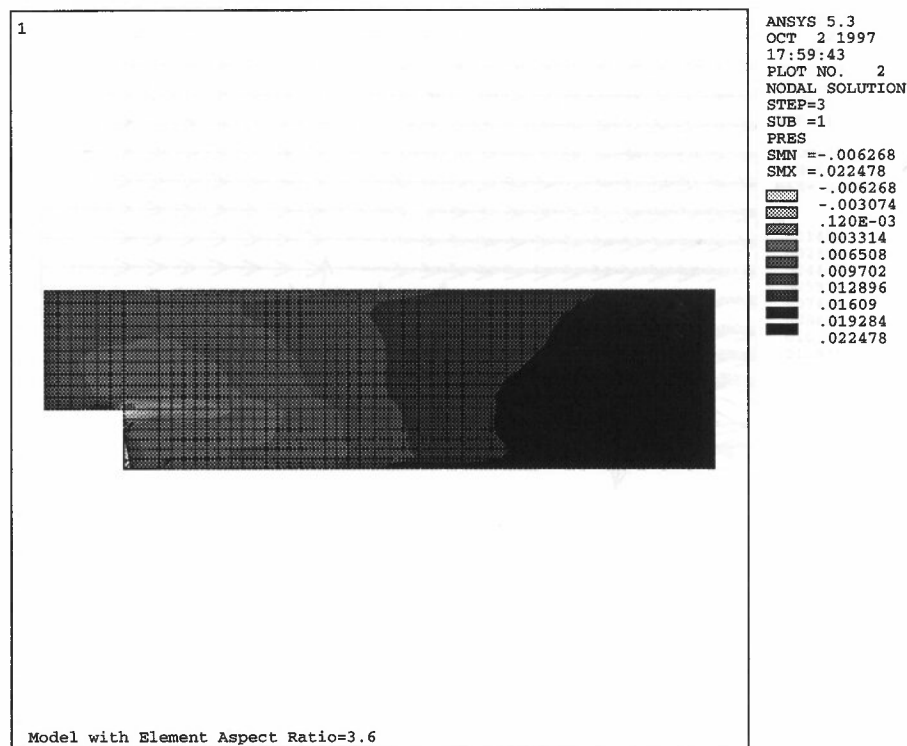


Figure 5.25: Contour Plot of Pressure Variation

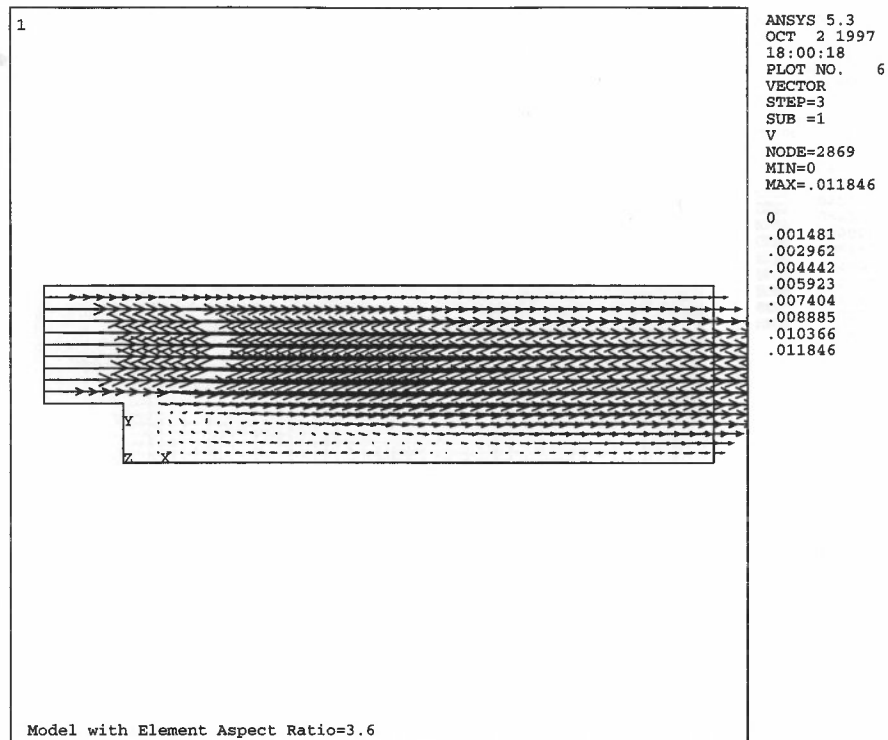


Figure 5.26: Vector Plot of Velocity Variation

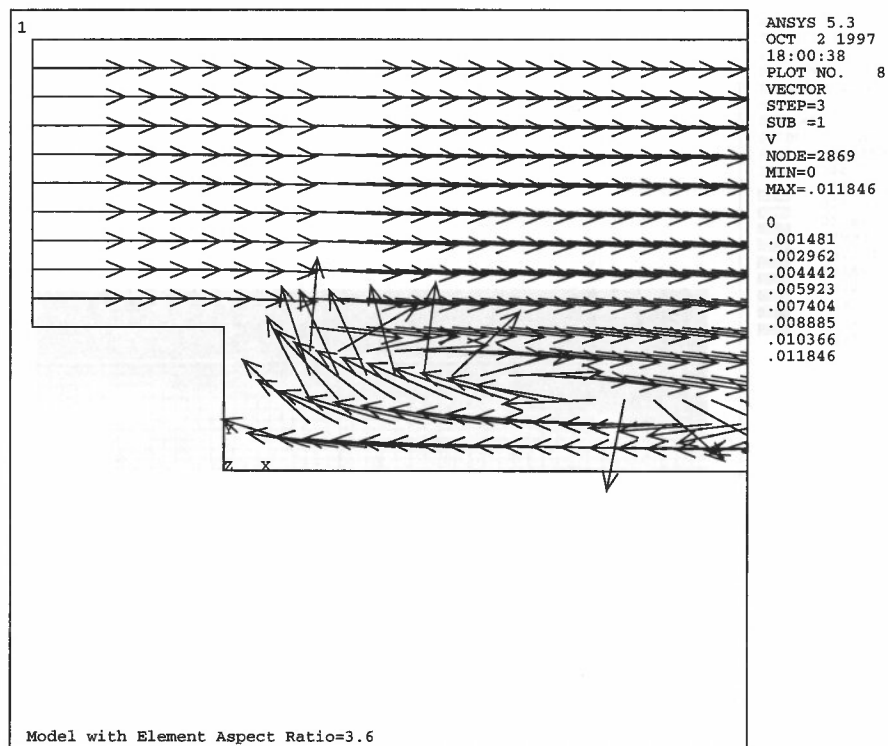


Figure 5.27: Magnification at Step location

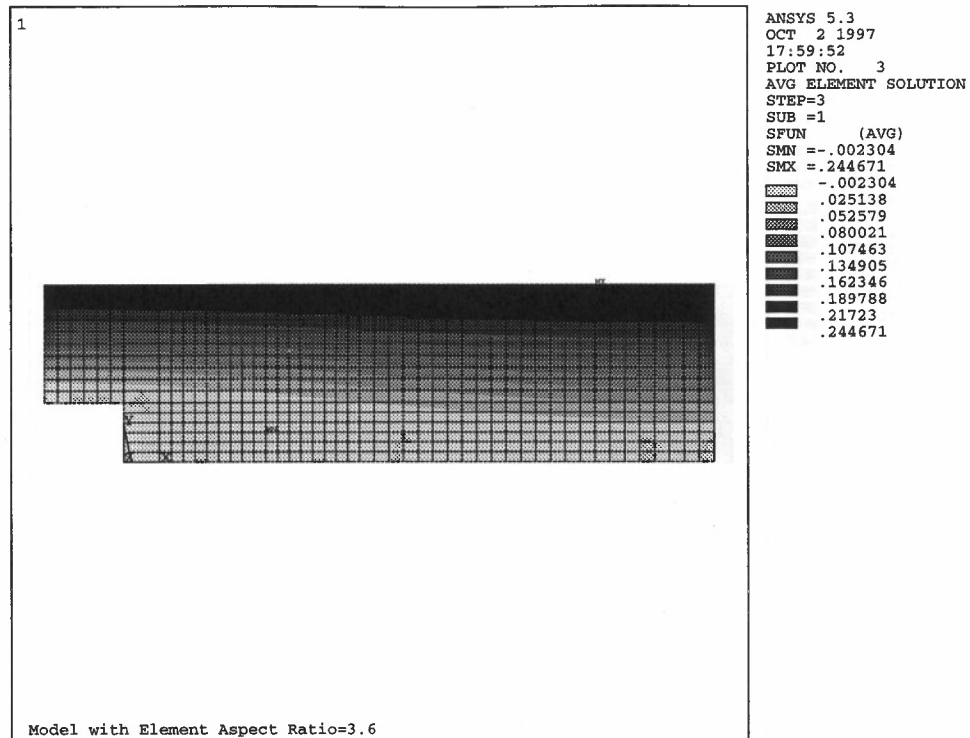


Figure 5.28: Contour Plot of Stream Function Variation; Averaged

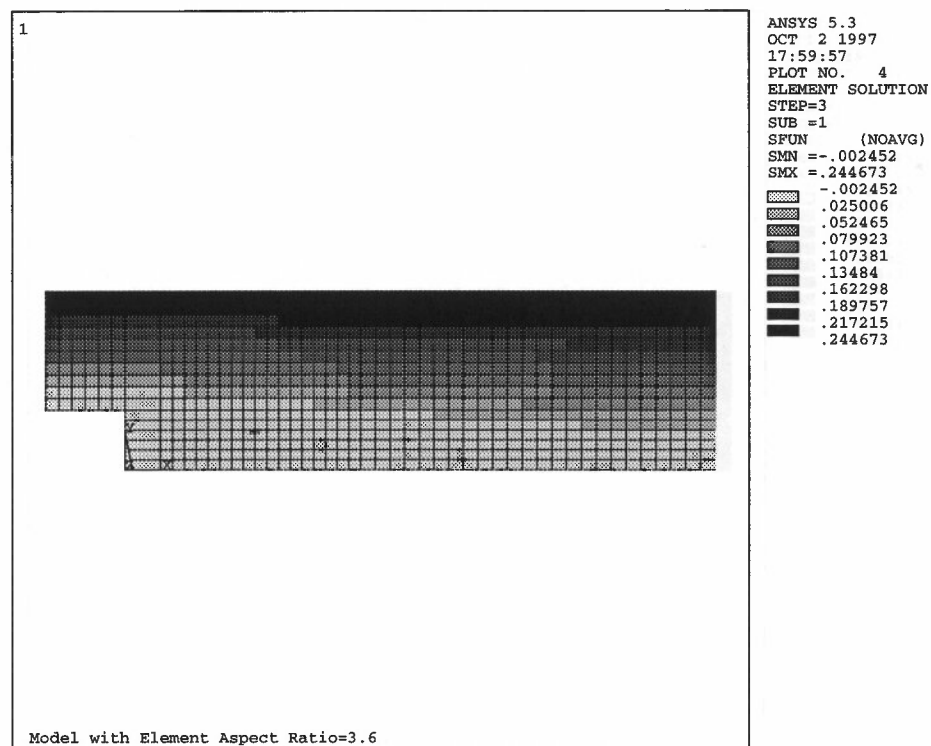


Figure 5.29: Contour Plot of Stream Function Variation; Non Averaged

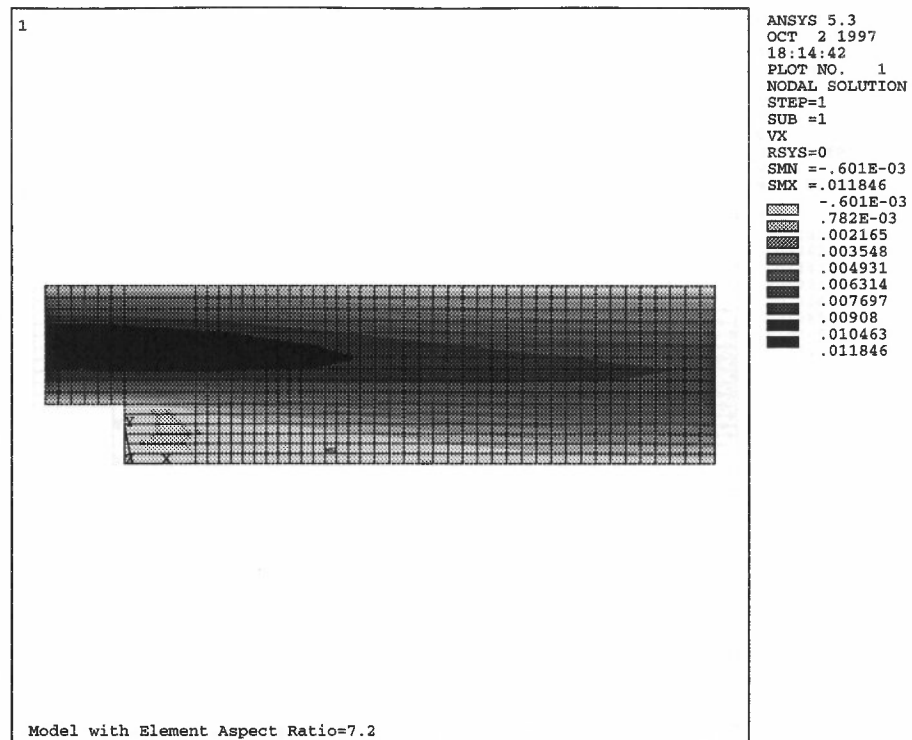


Figure 5.30: Contour Plot of Axial Velocity Variation

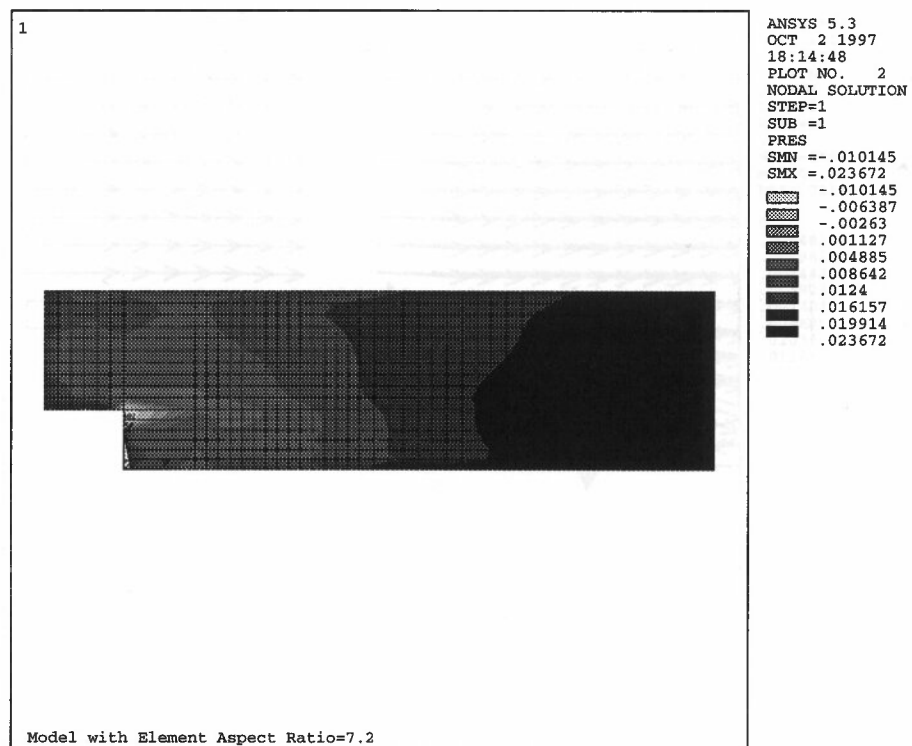


Figure 5.31: Contour Plot of Pressure Variation

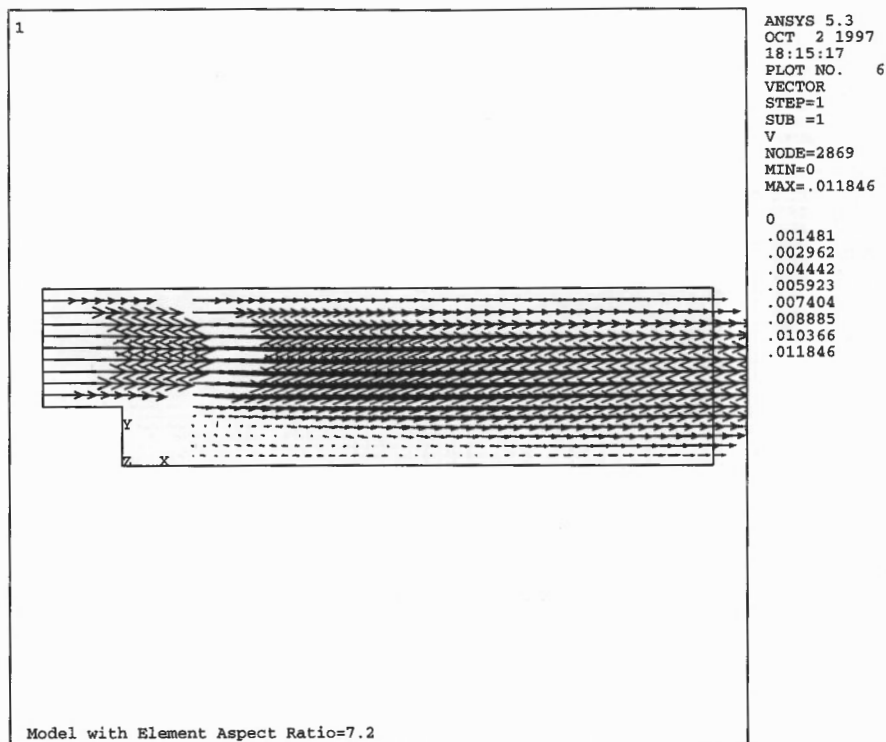


Figure 5.32: Vector Plot of Velocity Variation

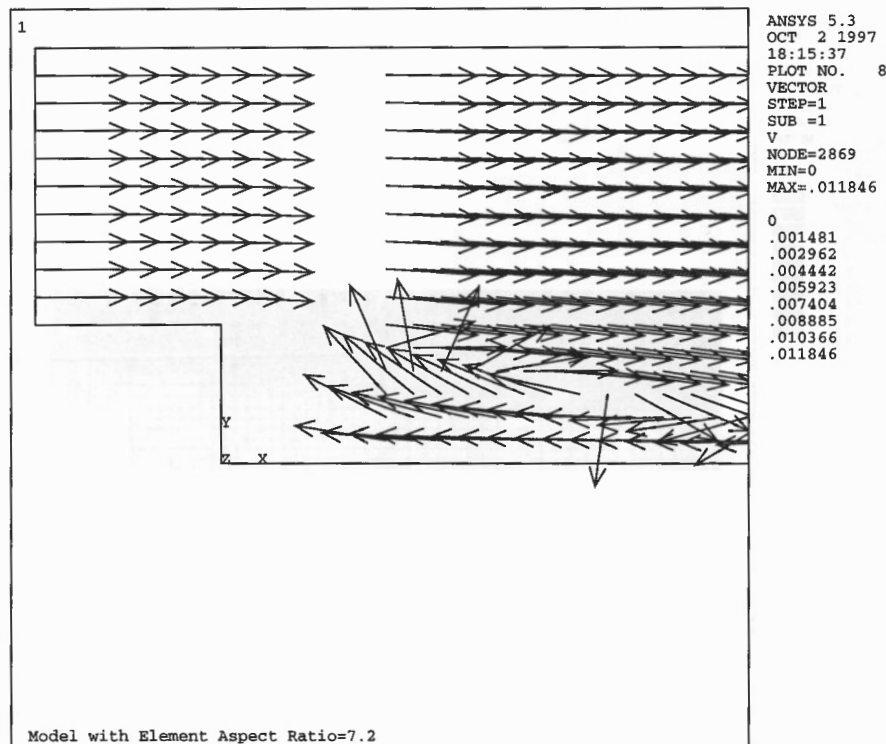


Figure 5.33: Magnification at Step location

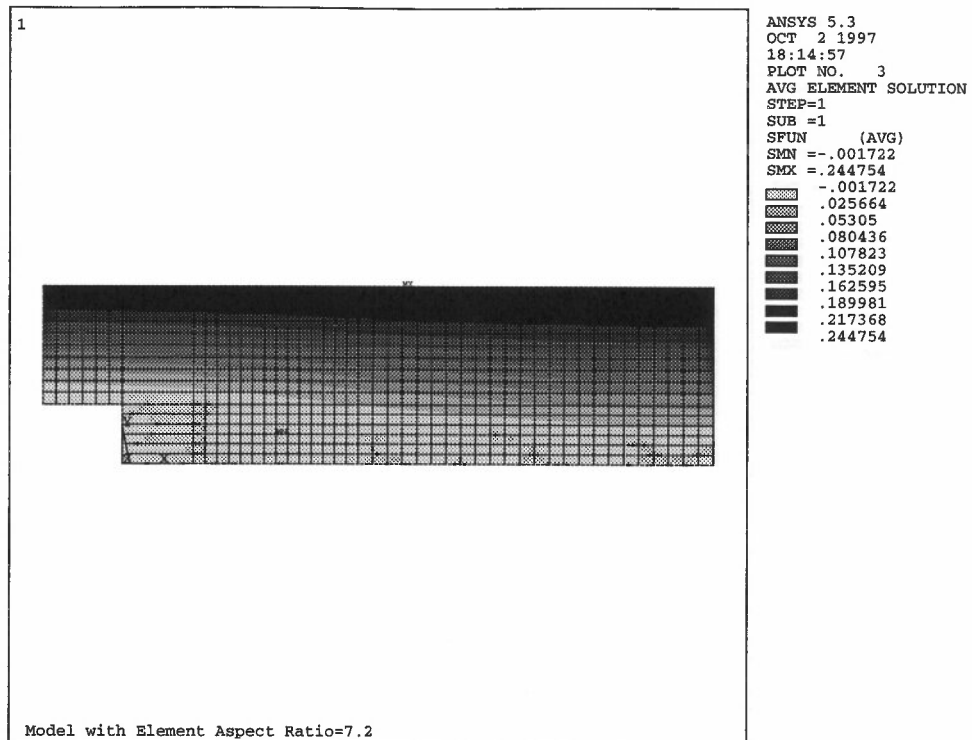


Figure 5.34: Contour Plot of Stream Function Variation; Averaged

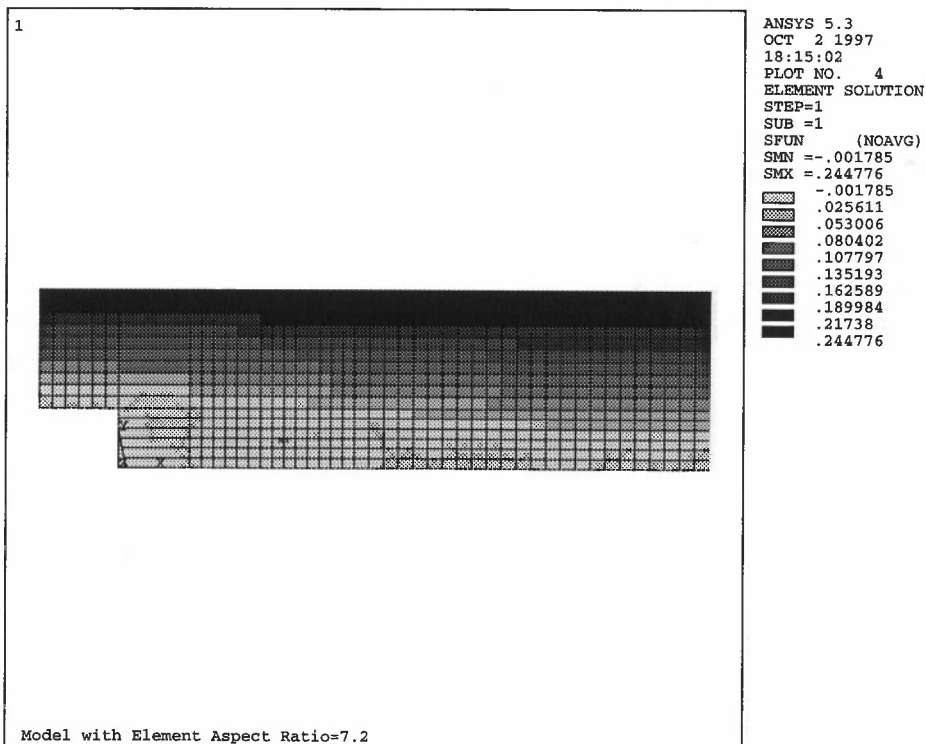


Figure 5.35: Contour Plot of Stream Function Variation; Non Averaged

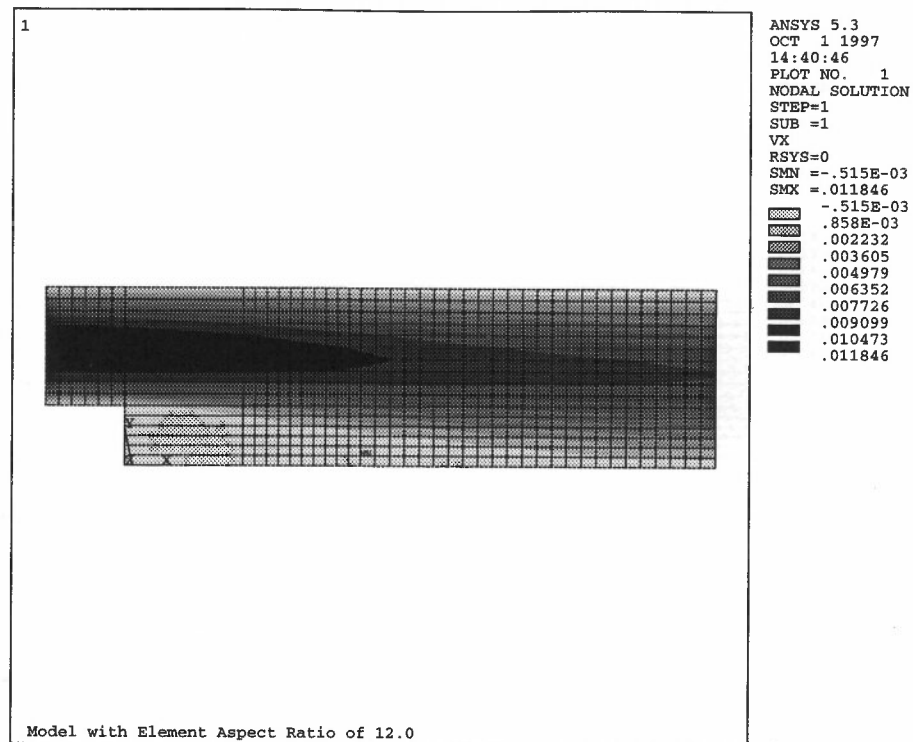


Figure 5.36: Contour Plot of Axial Velocity Variation

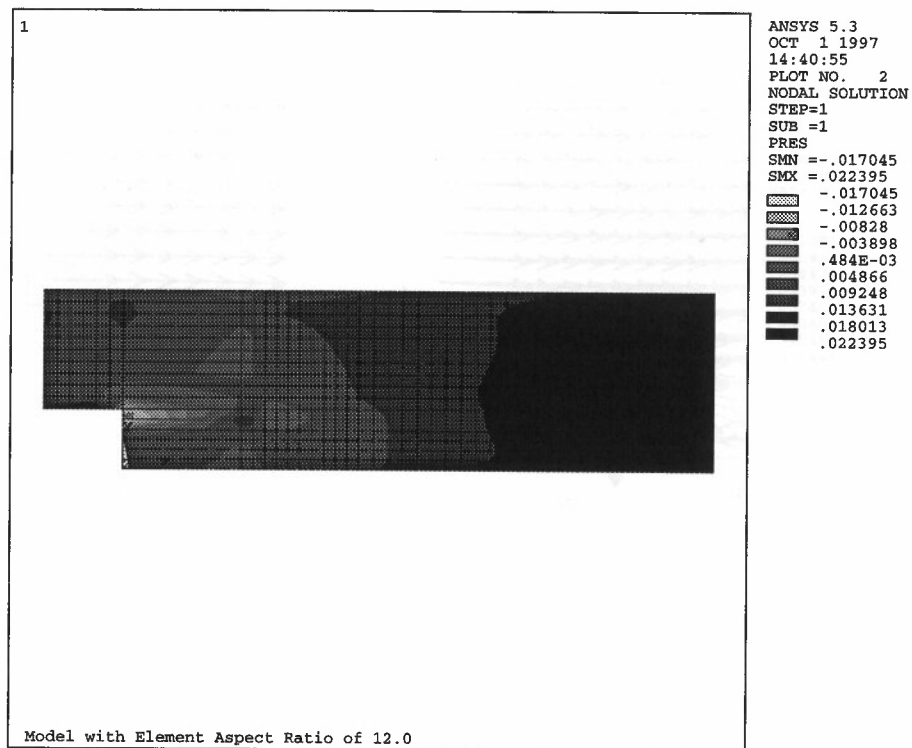


Figure 5.37: Contour Plot of Pressure Variation

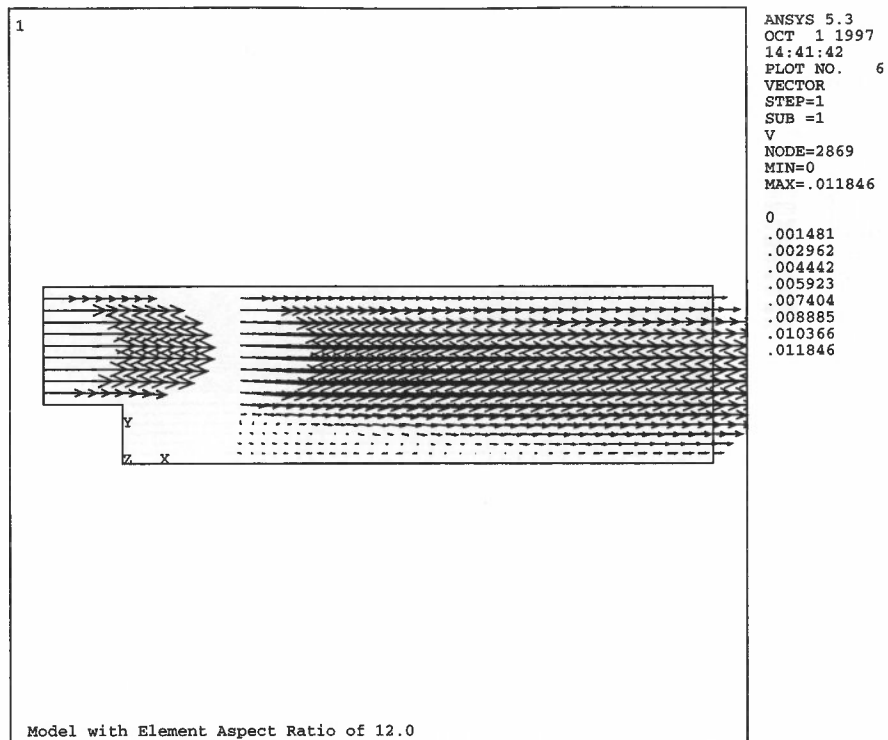


Figure 5.38: Vector Plot of Velocity Variation

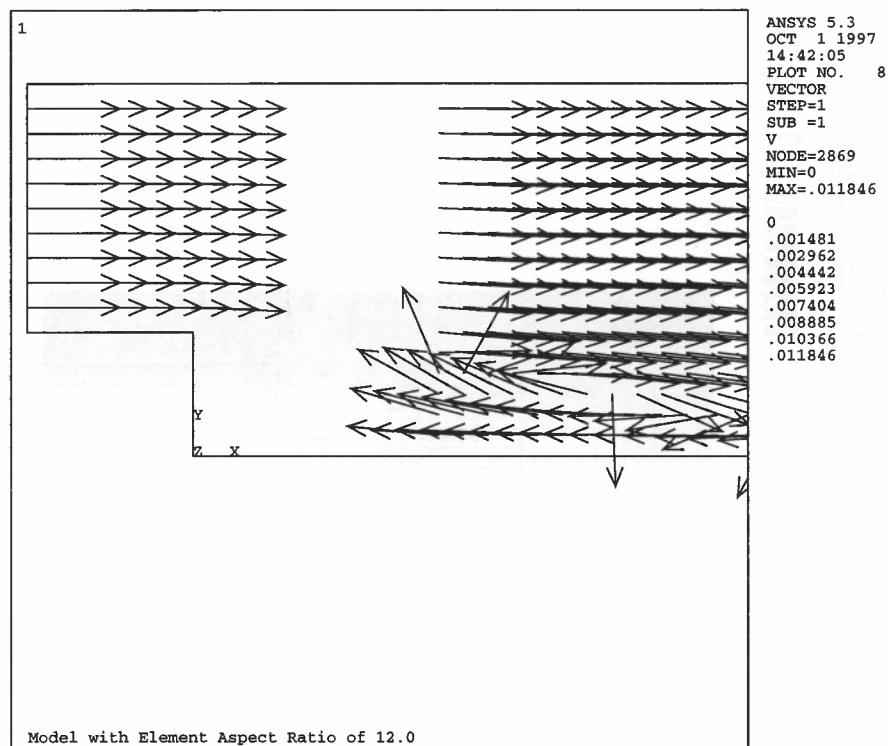


Figure 5.39: Magnification at Step location

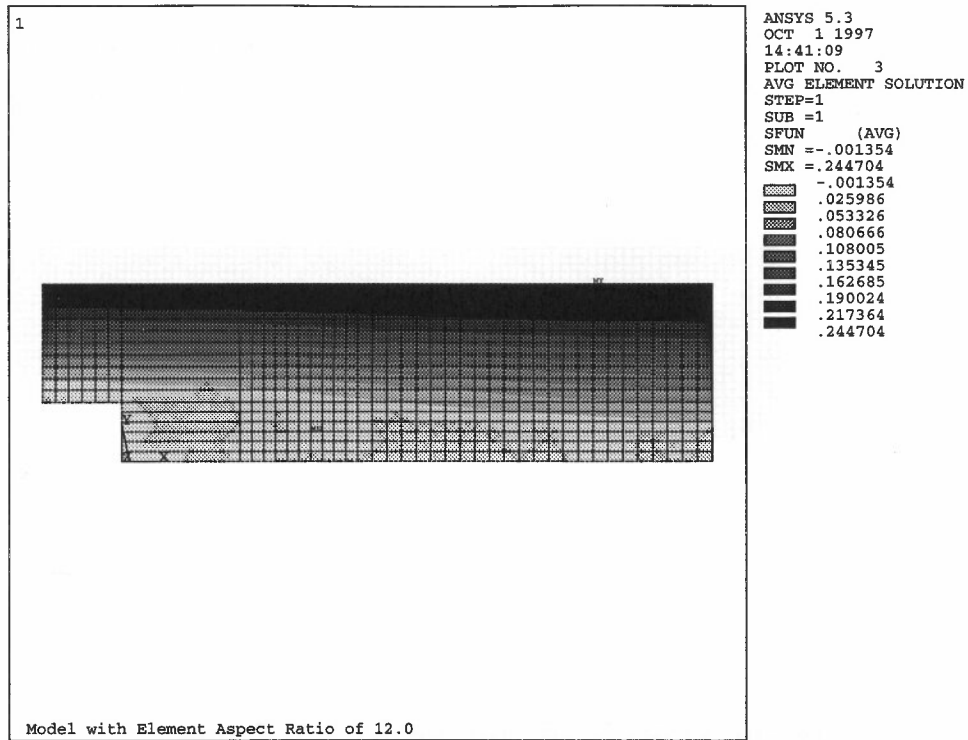


Figure 5.40: Contour Plot of Stream Function Variation; Averaged

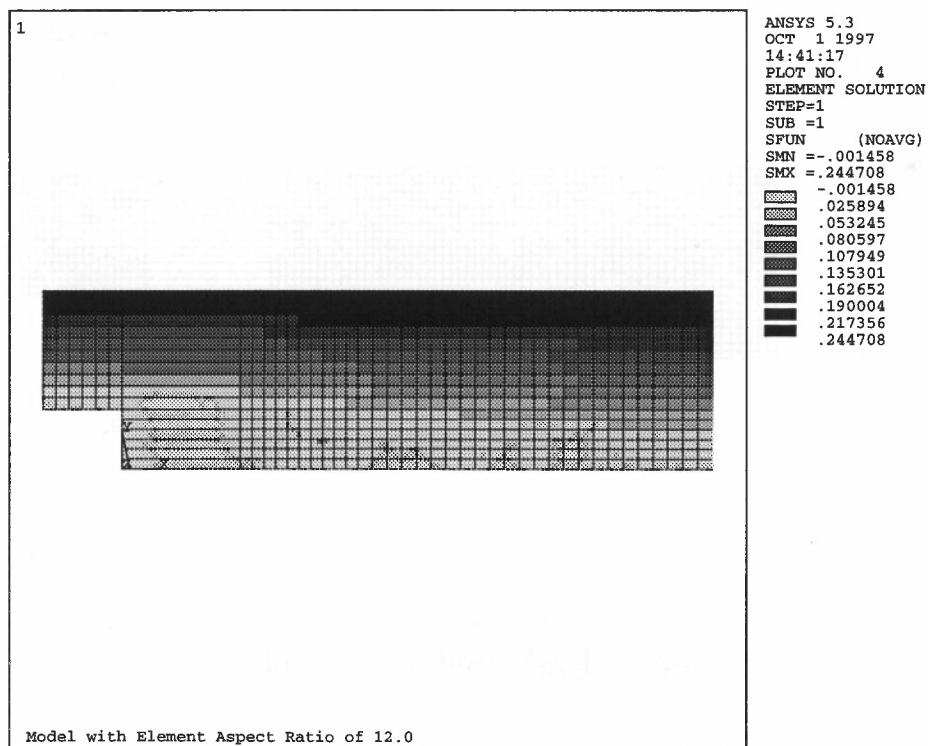


Figure 5.41: Contour Plot of Stream Function Variation; Non Averaged

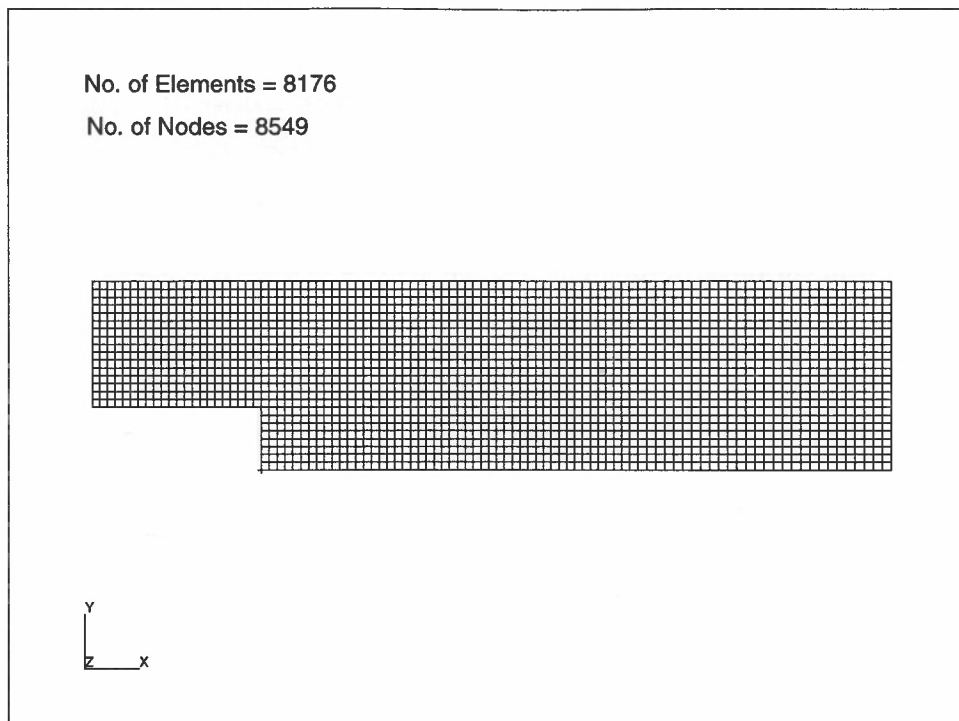


Figure 5.42: Modified Mesh for Gradient Analysis

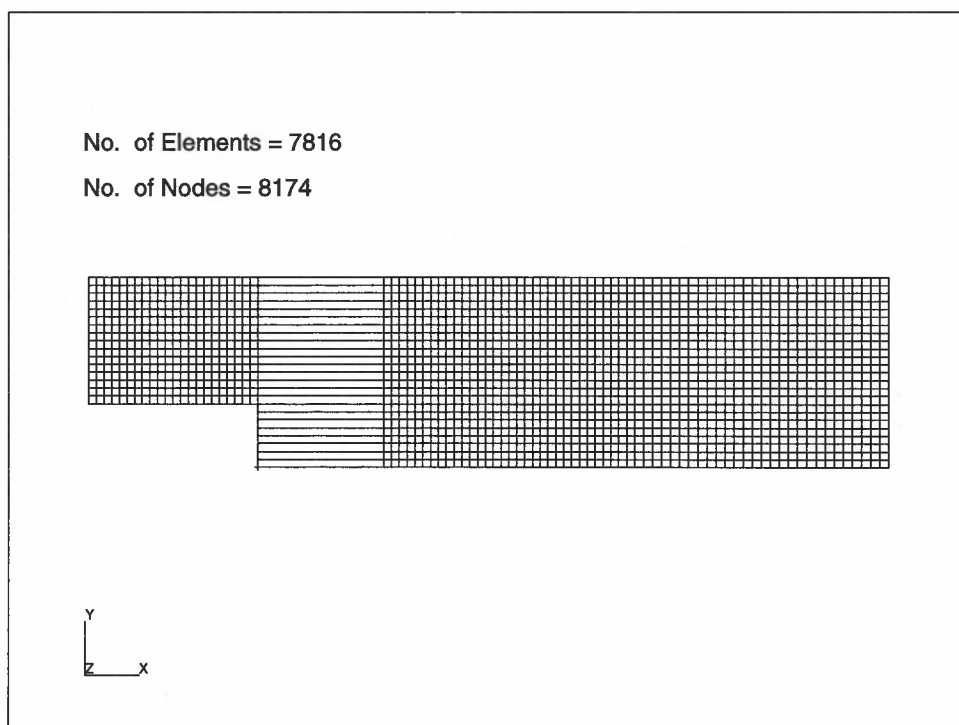


Figure 5.43: Steep Gradient Mesh

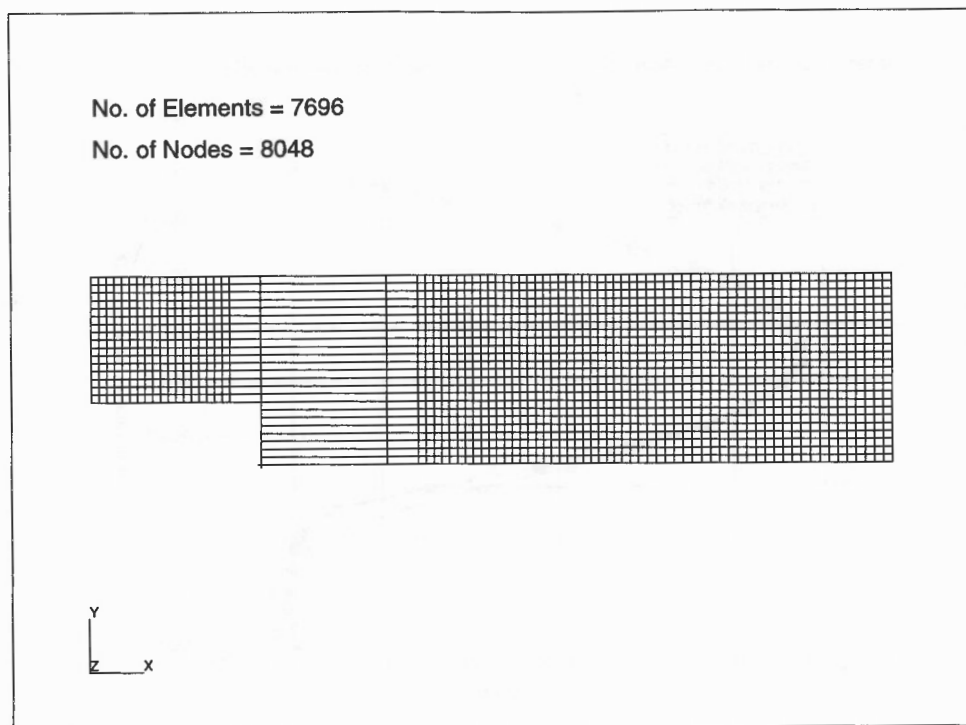


Figure 5.44: Medium Gradient Mesh

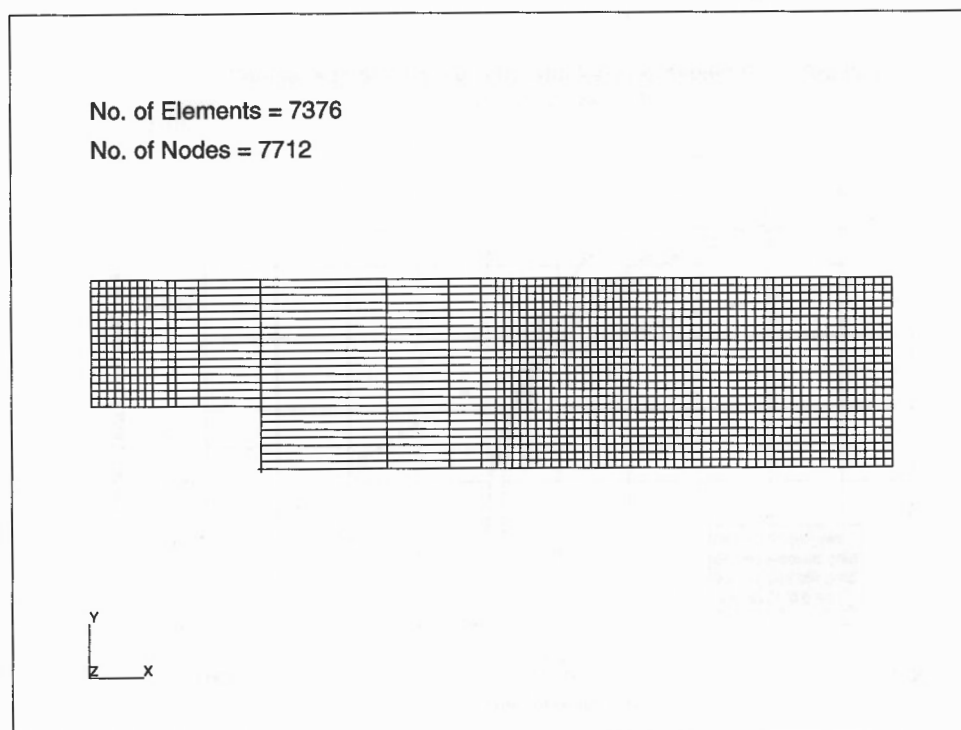


Figure 5.45: Smooth Gradient Mesh

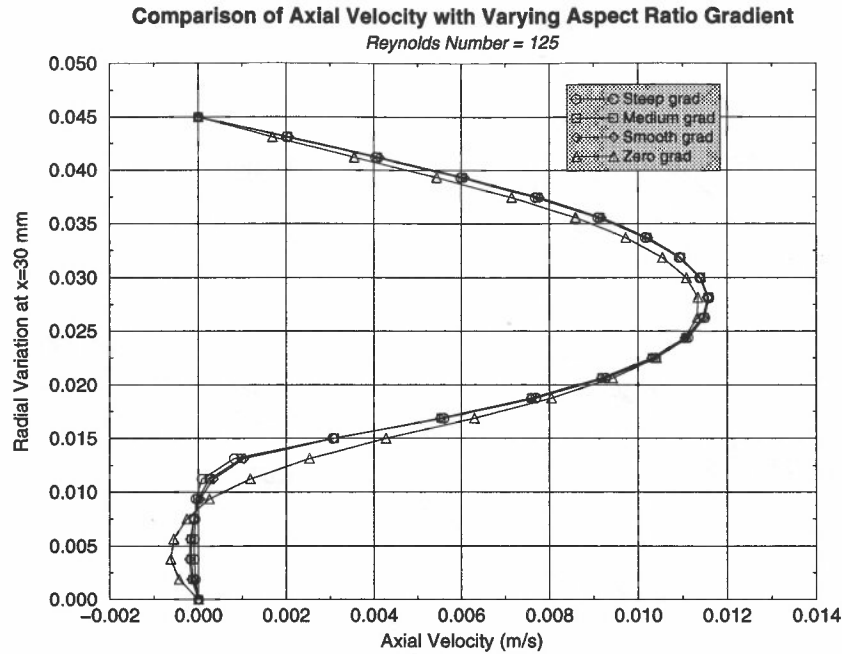


Figure 5.46: Velocity Profile at Varying Aspect Ratio Gradient

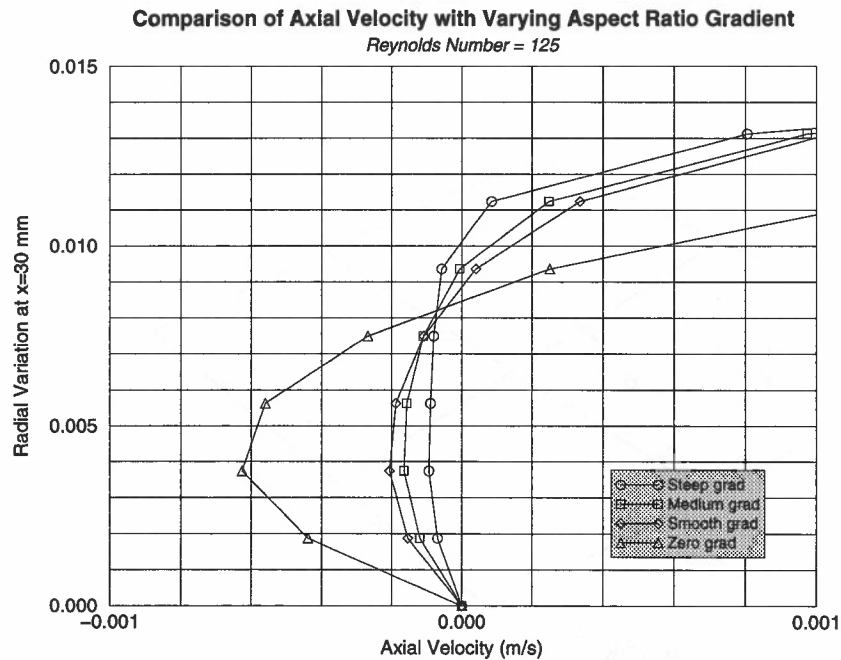


Figure 5.47: Magnification Plot at Step Location

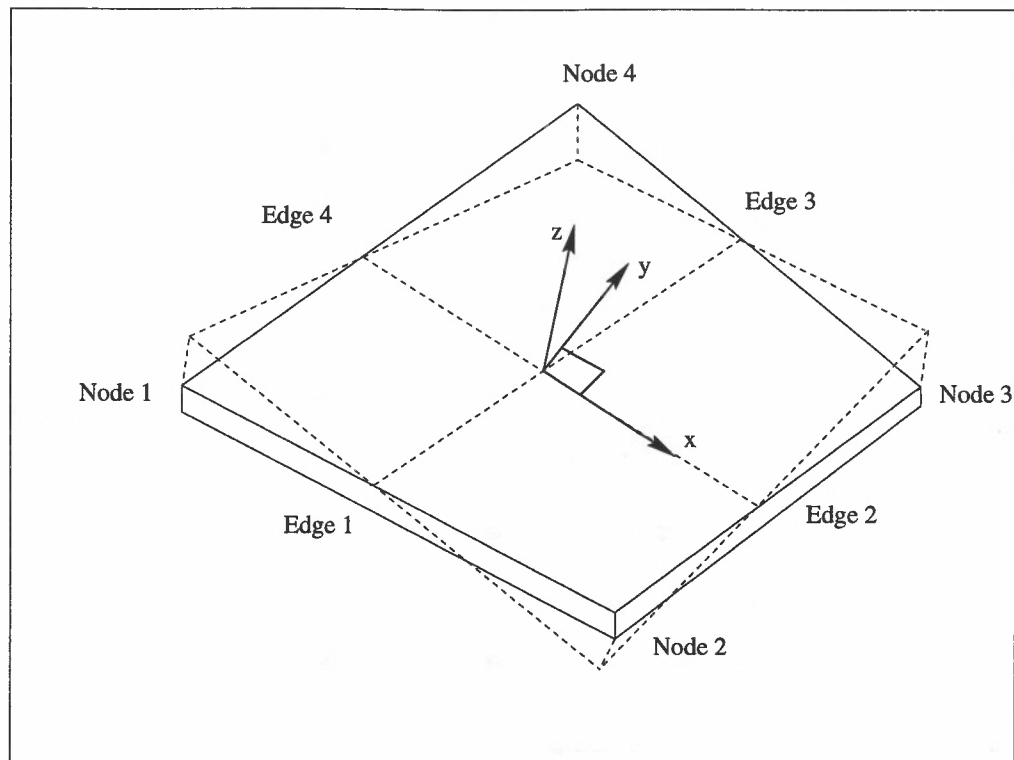


Figure 5.48: Quad Skew Angle Element Definition

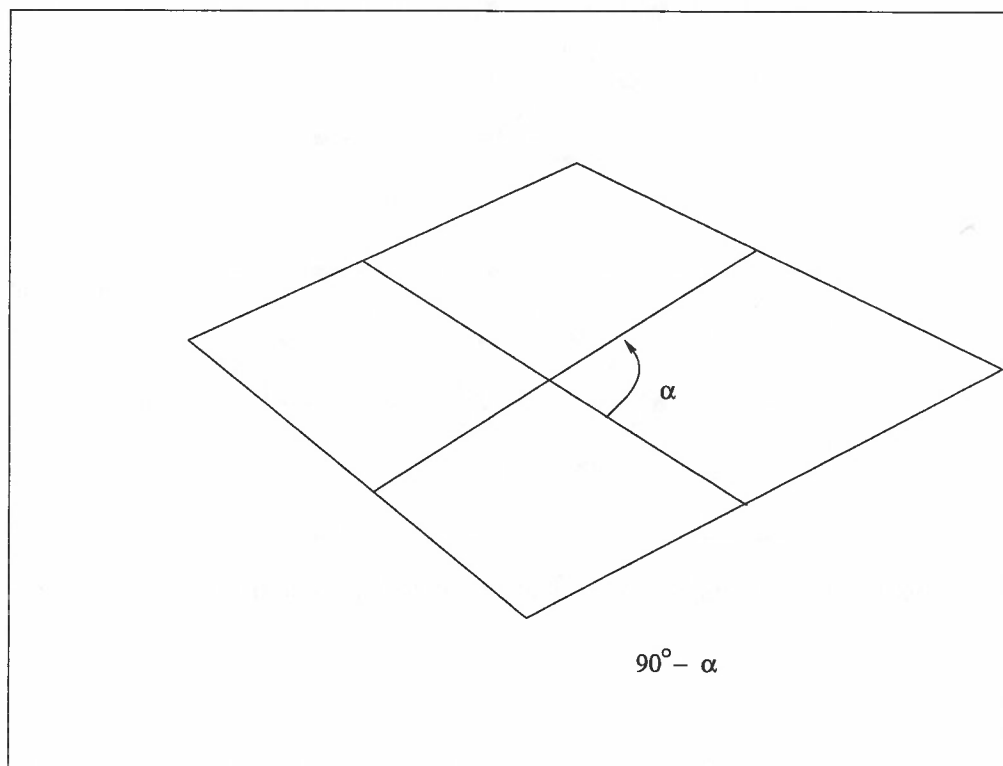


Figure 5.49: Quad Skew Angle Element Normalisation Definition

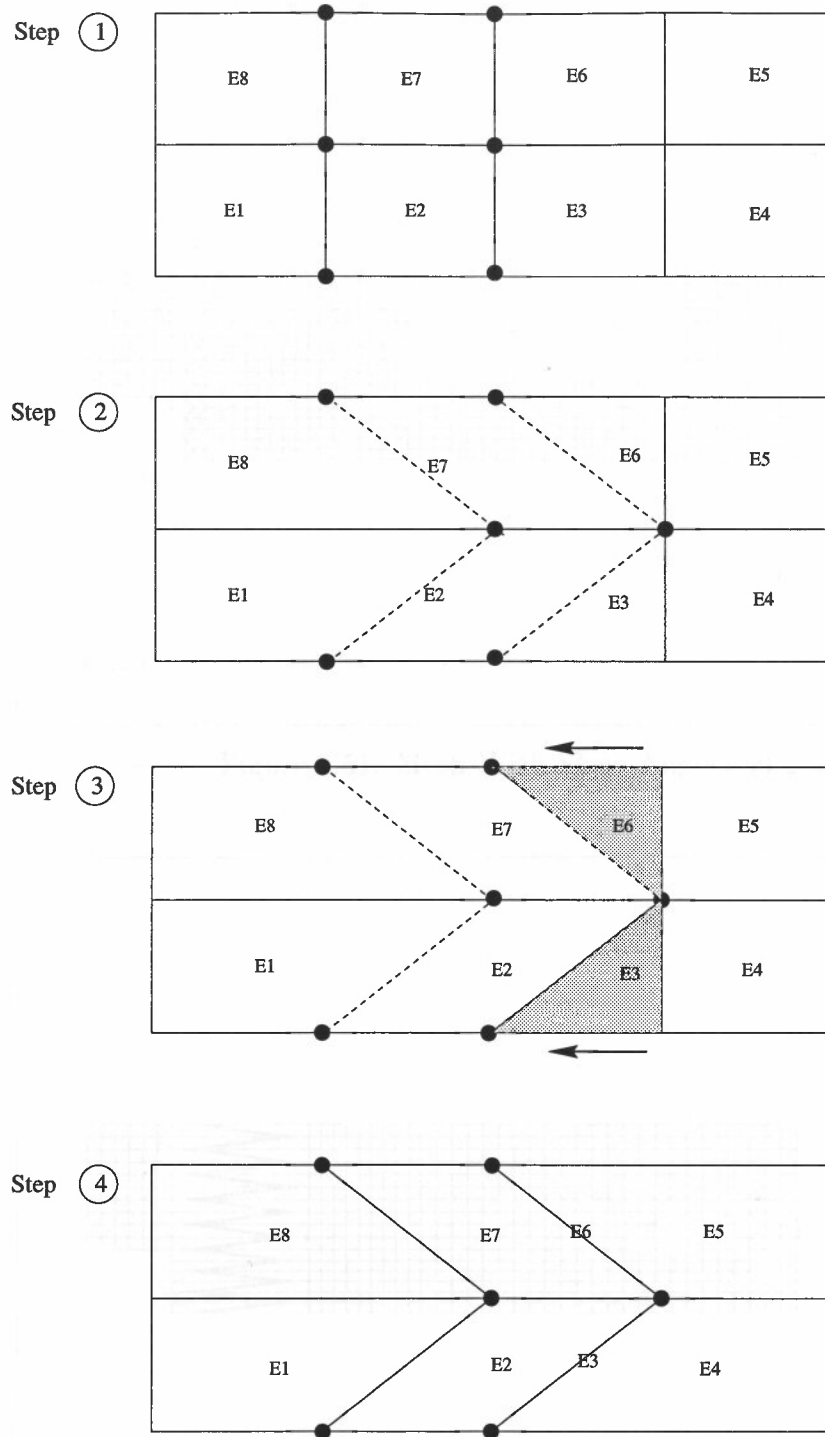


Figure 5.50: Illustration of Element Skew Methodology

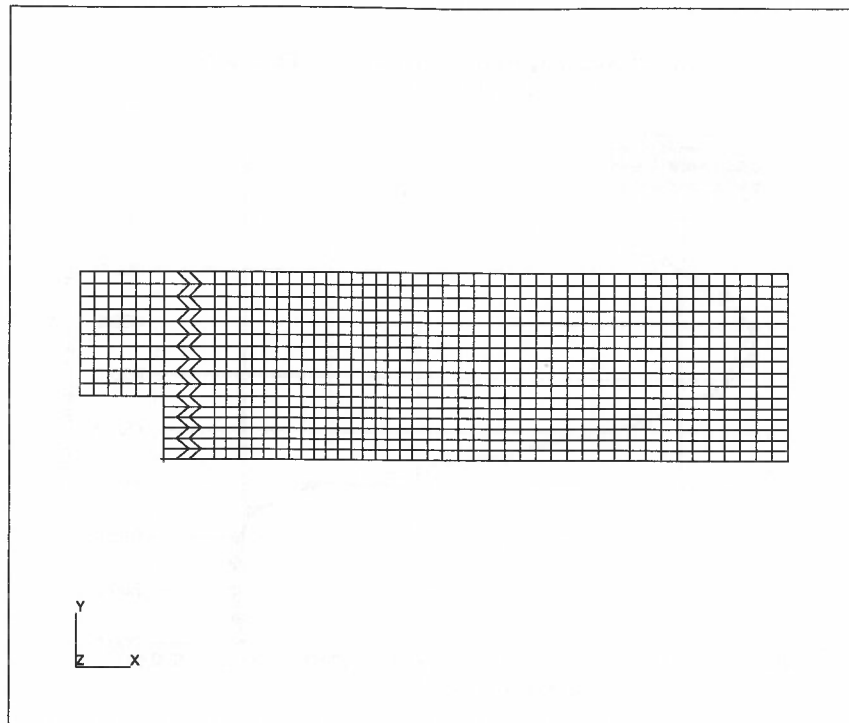


Figure 5.51: Mesh With Skew Angle=50.2 degrees

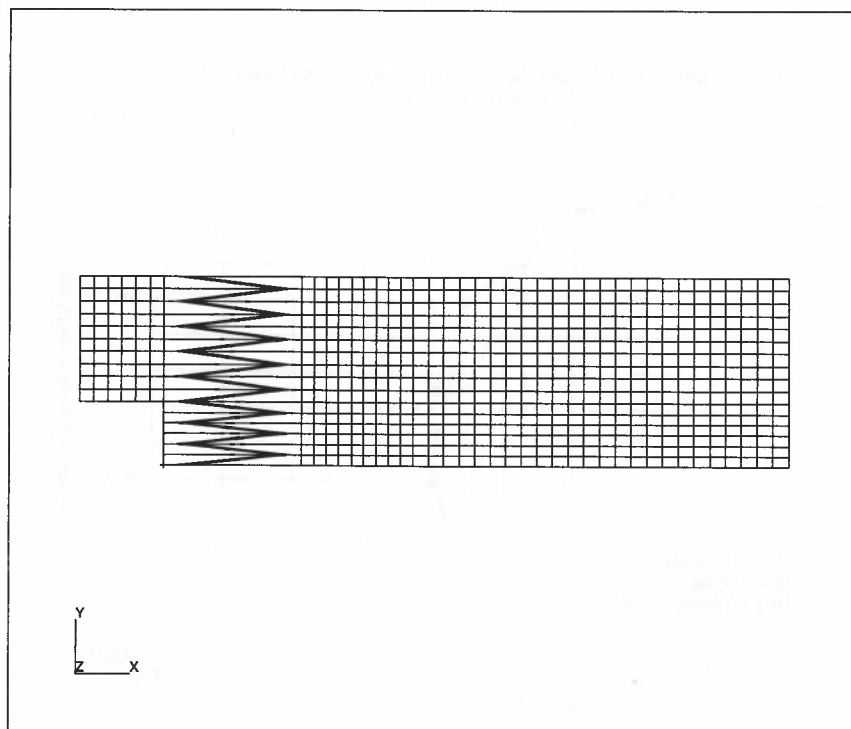


Figure 5.52: Mesh With Skew Angle=84.1 degrees

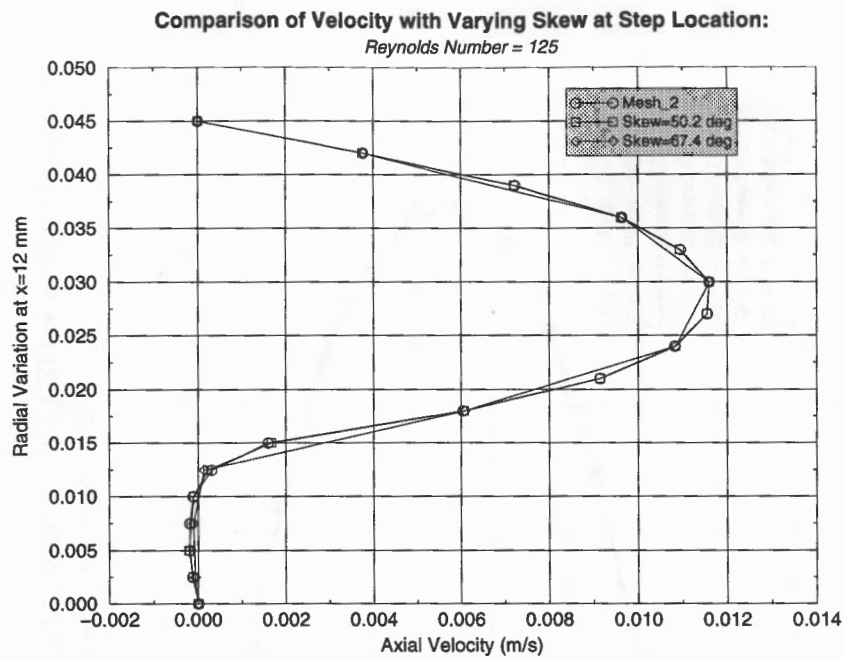


Figure 5.53: Skew Angle Variation to Inspection Point One

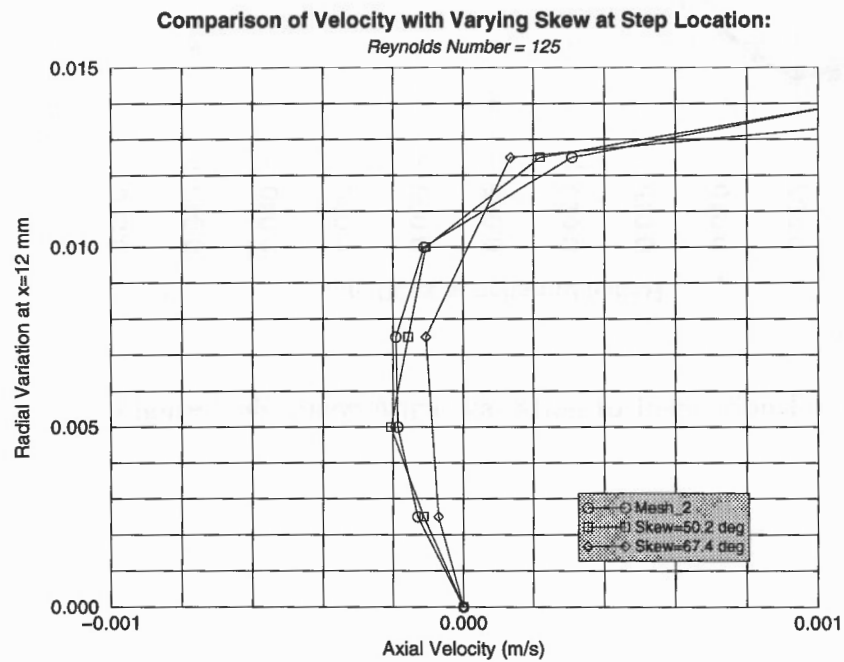


Figure 5.54: Magnification Plot at Step Location

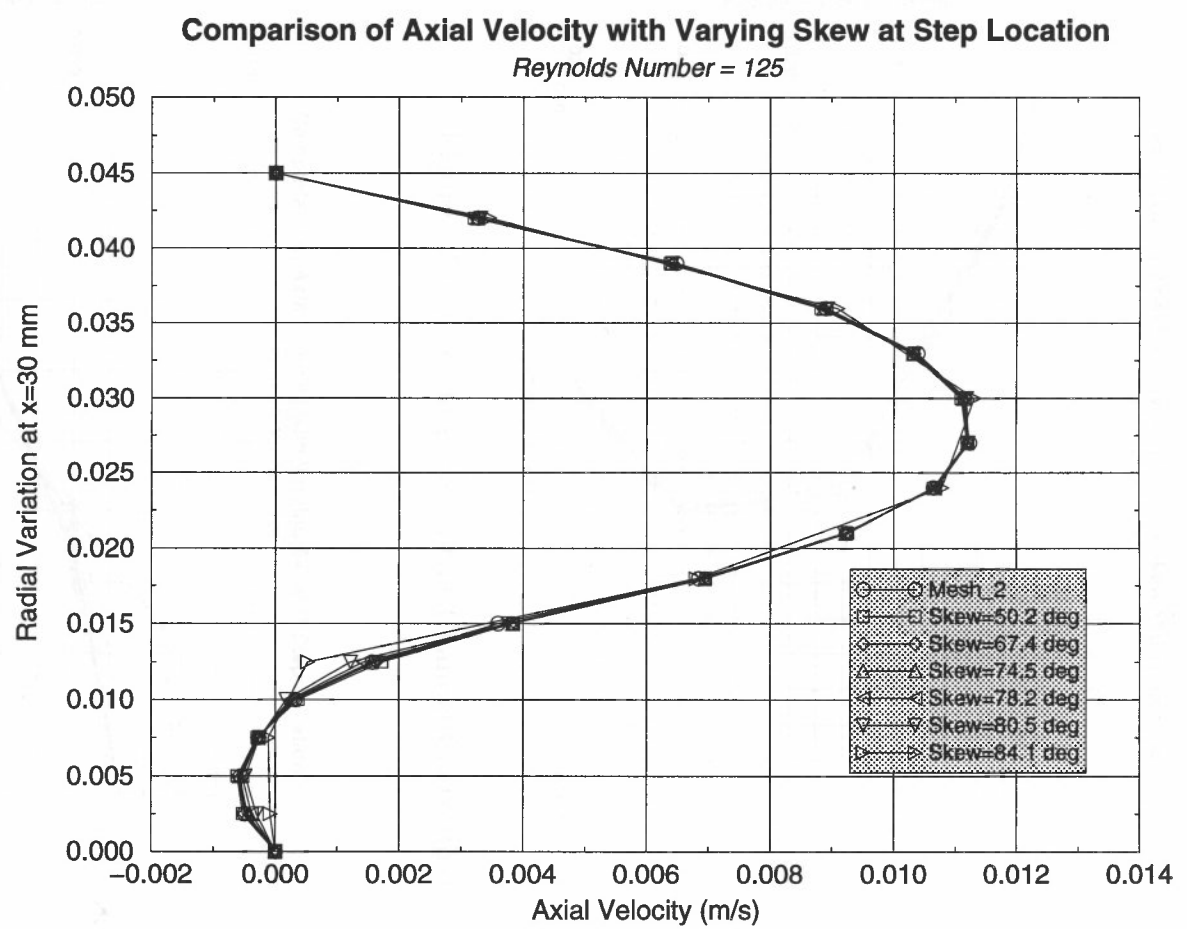


Figure 5.55: Skew Angle Variation to Inspection Point Two

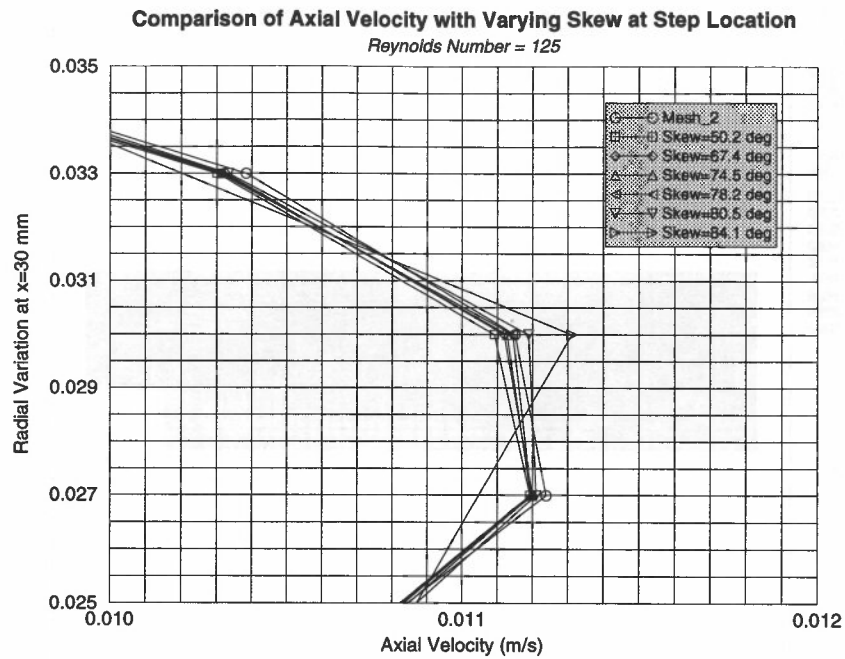


Figure 5.56: Magnification Plot at Channel Centre-line

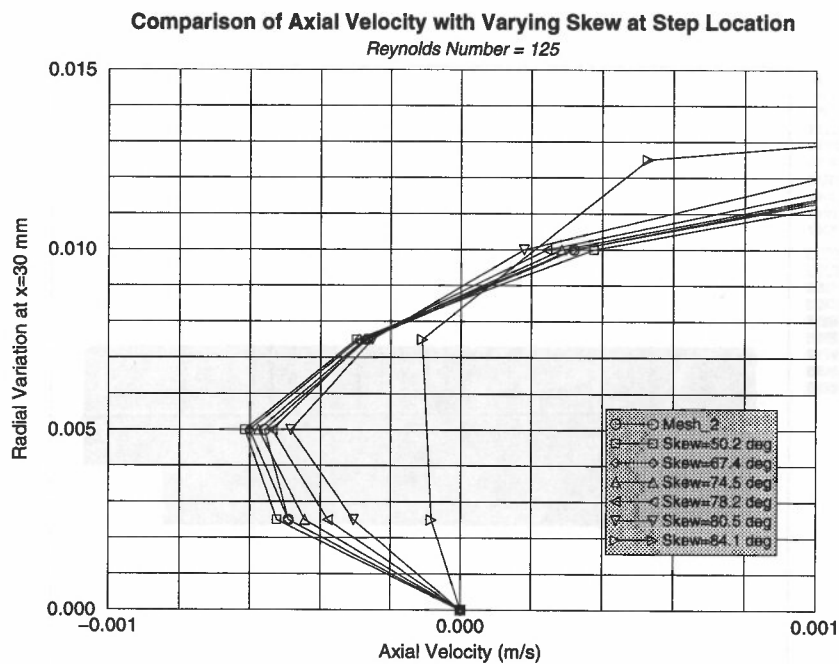


Figure 5.57: Magnification Plot at Step Location

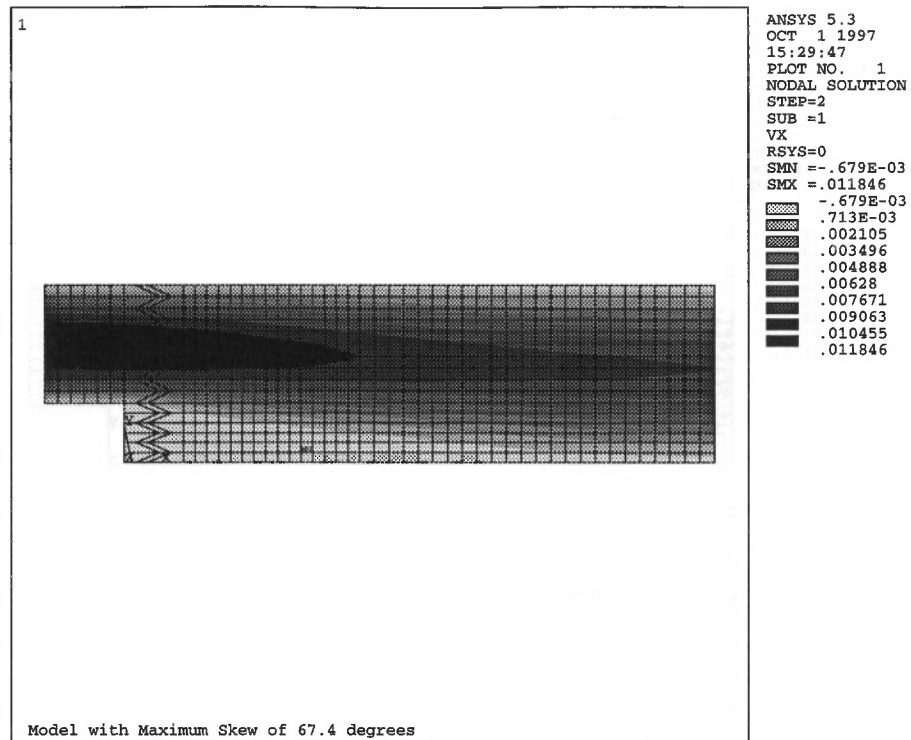


Figure 5.58: Contour Plot of Axial Velocity Variation

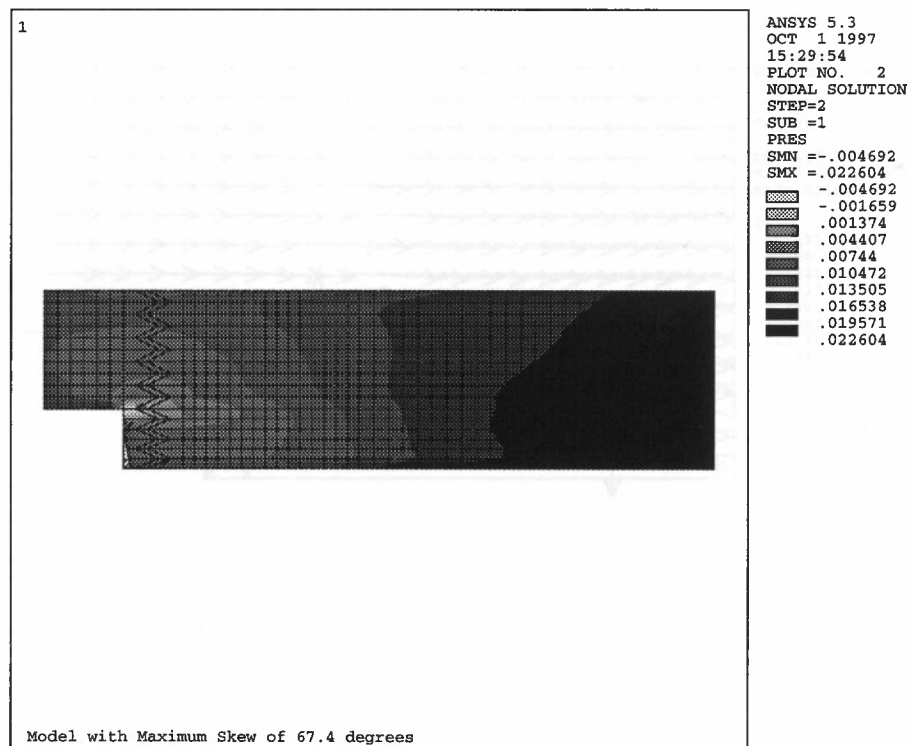


Figure 5.59: Contour Plot of Pressure Variation

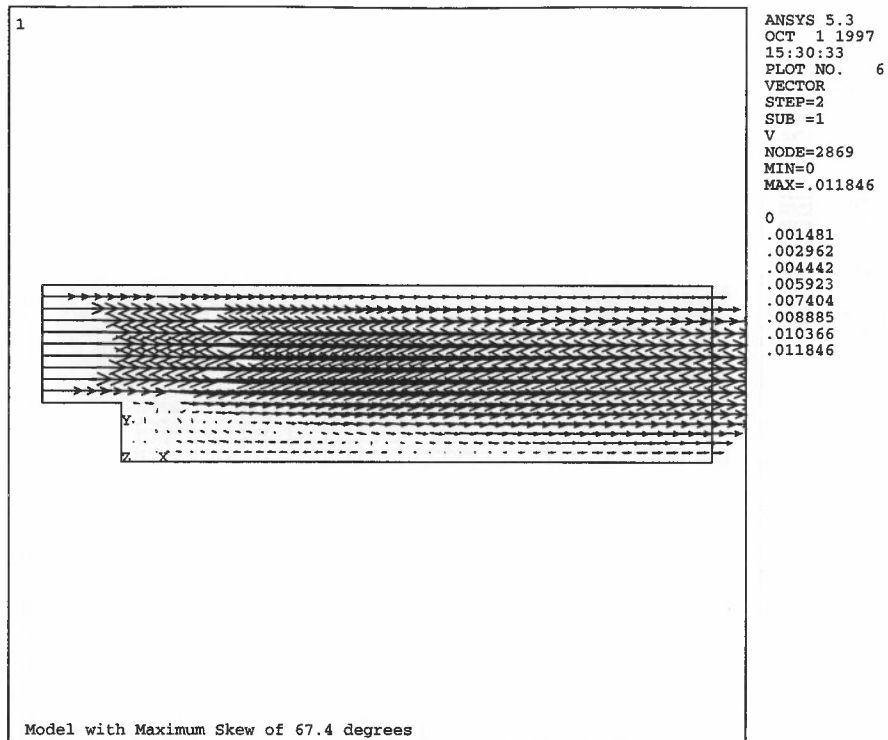


Figure 5.60: Vector Plot of Velocity Variation

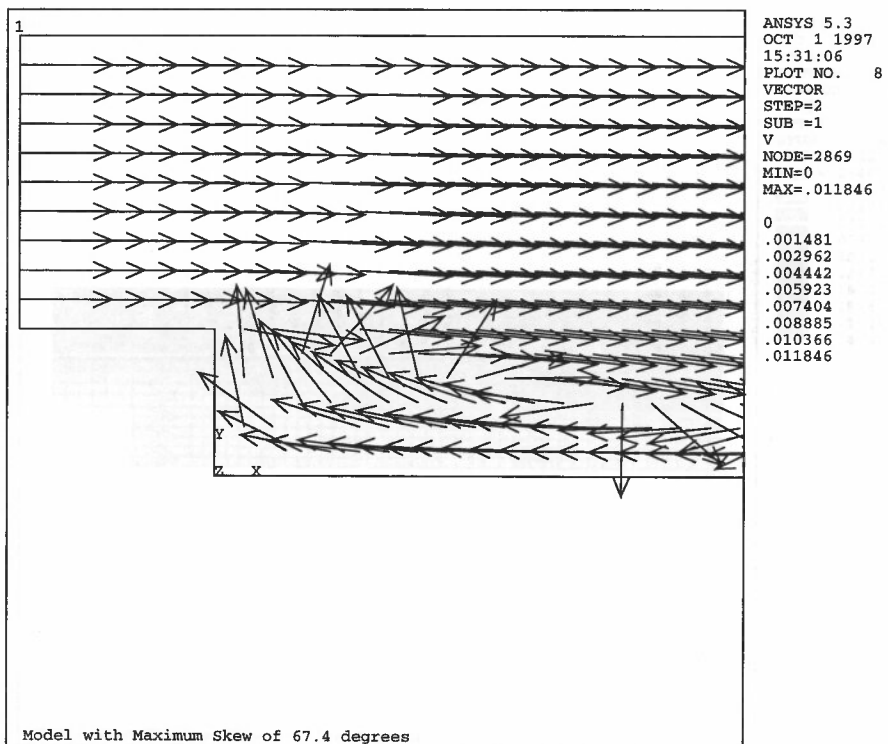


Figure 5.61: Magnification at Step location

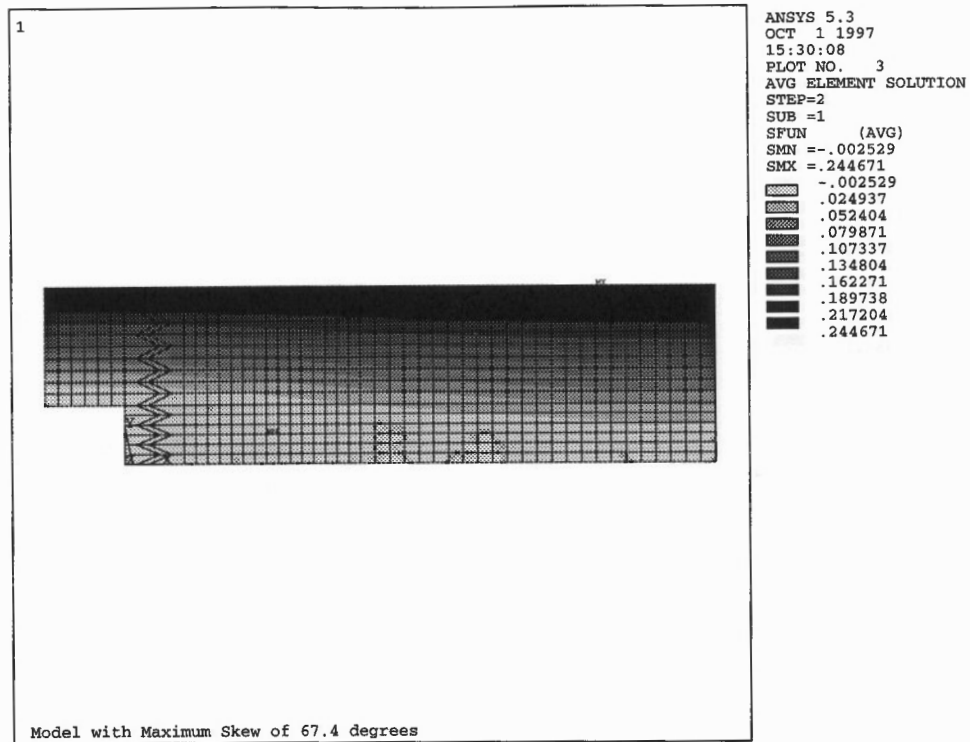


Figure 5.62: Contour Plot of Stream Function Variation; Averaged

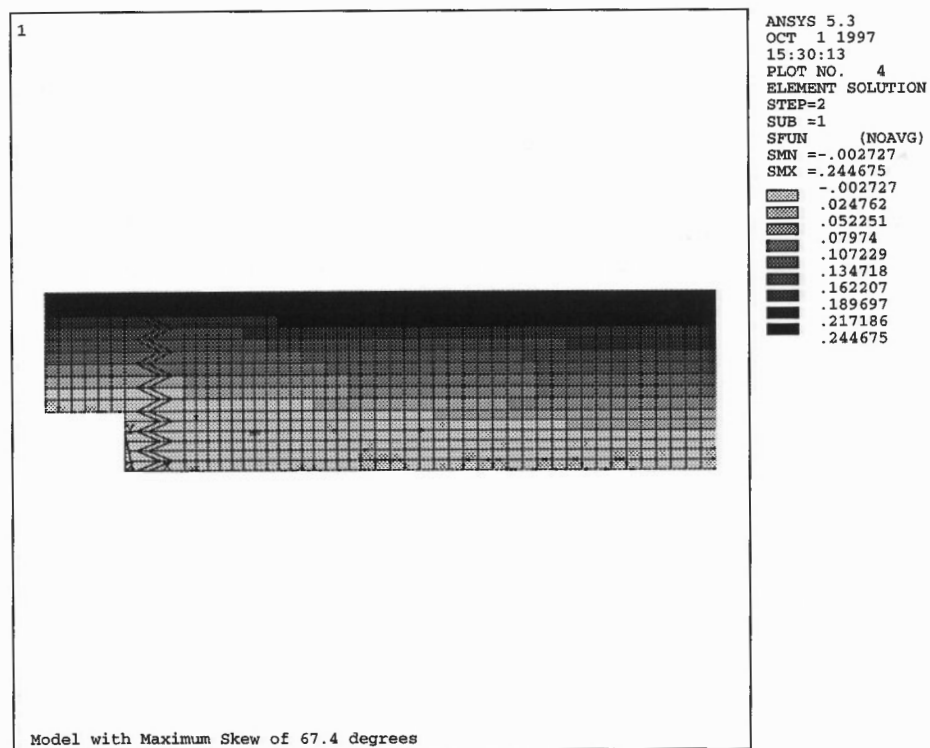


Figure 5.63: Contour Plot of Stream Function Variation; Non Averaged

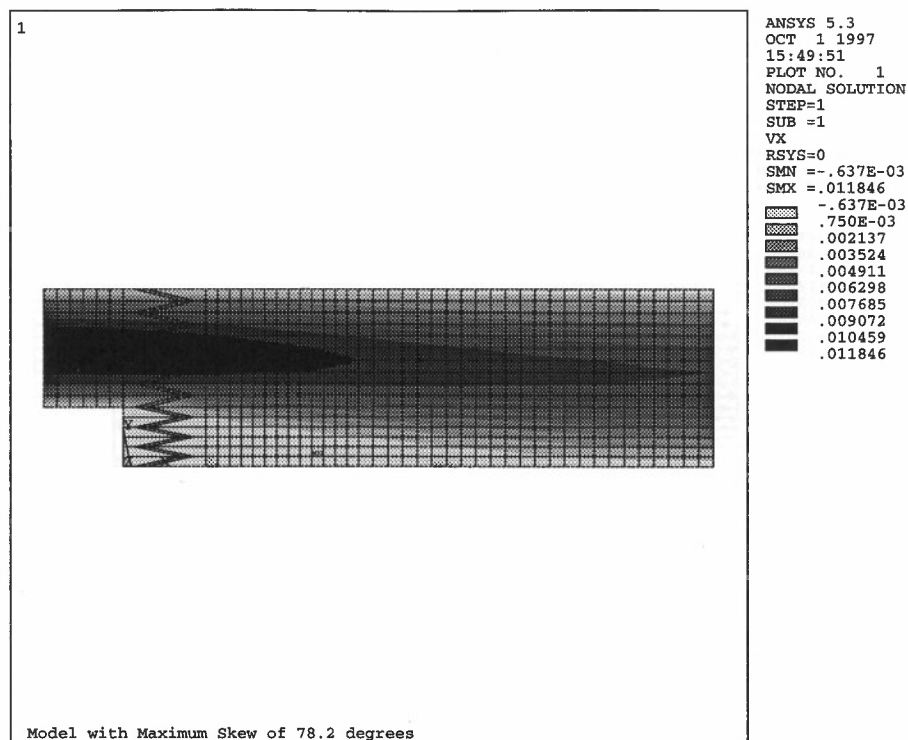


Figure 5.64: Contour Plot of Axial Velocity Variation

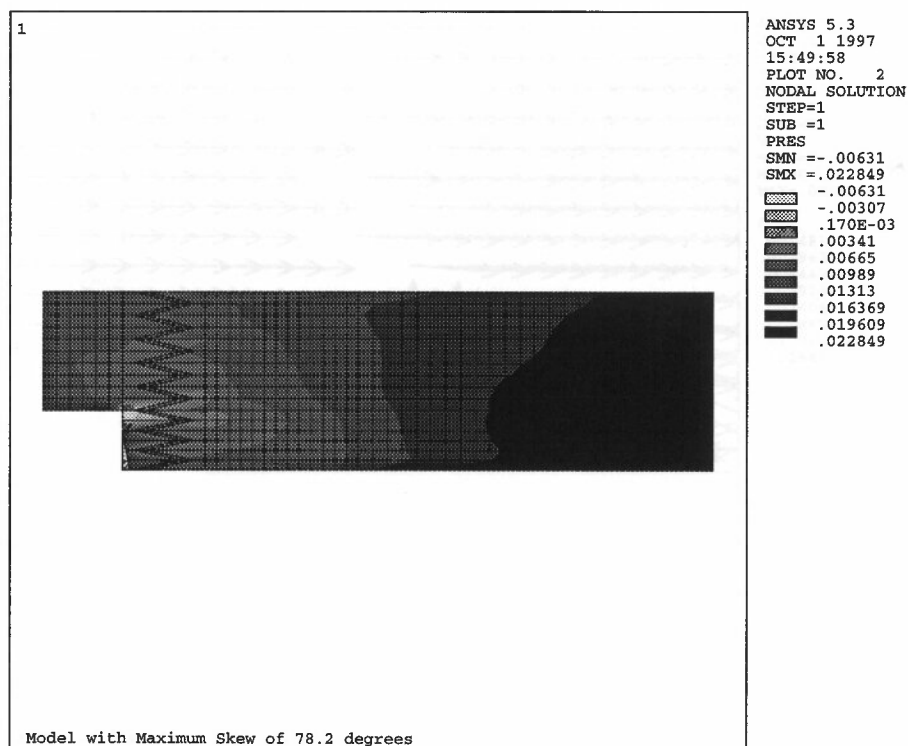


Figure 5.65: Contour Plot of Pressure Variation

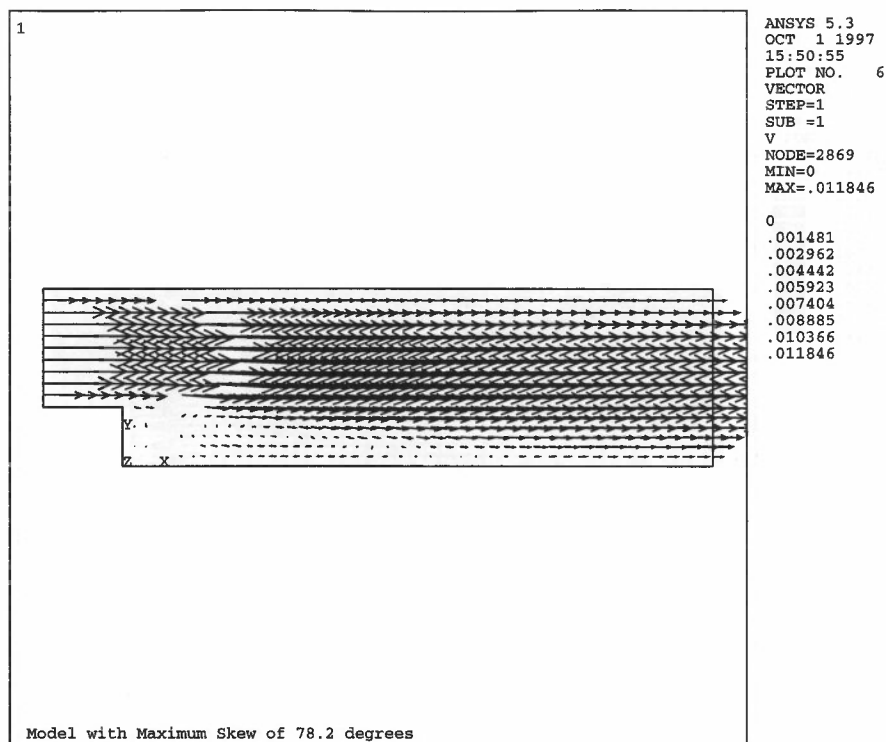


Figure 5.66: Vector Plot of Velocity Variation

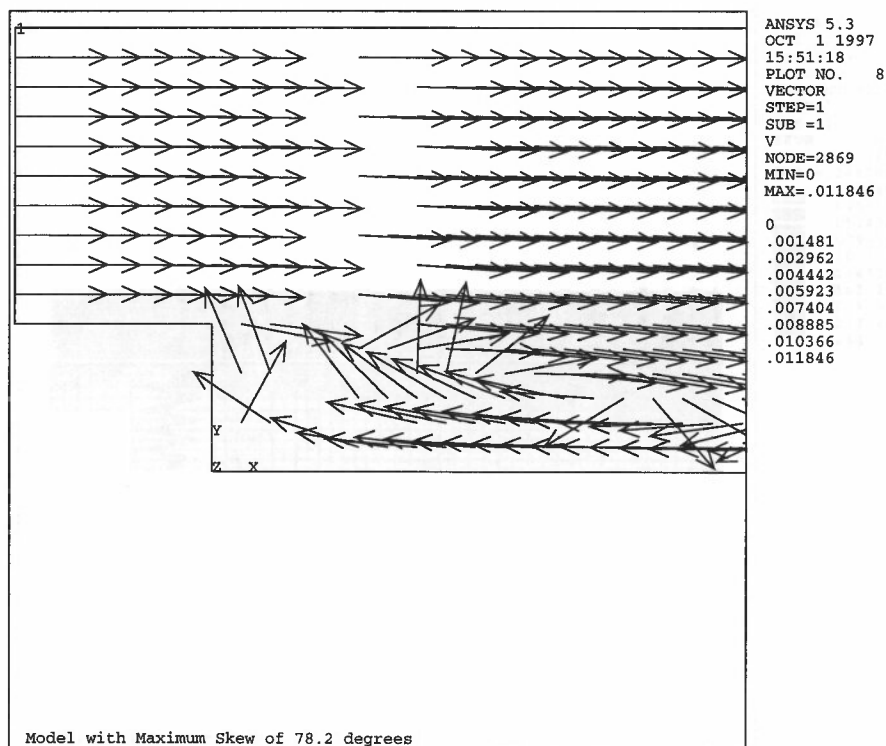


Figure 5.67: Magnification at Step location

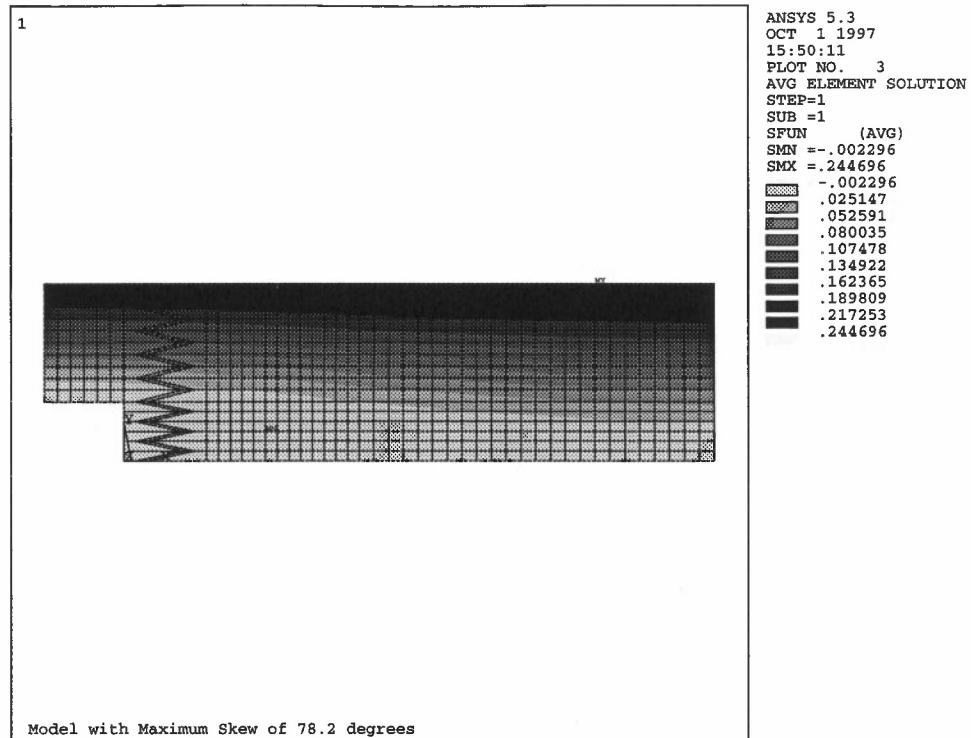


Figure 5.68: Contour Plot of Stream Function Variation; Averaged

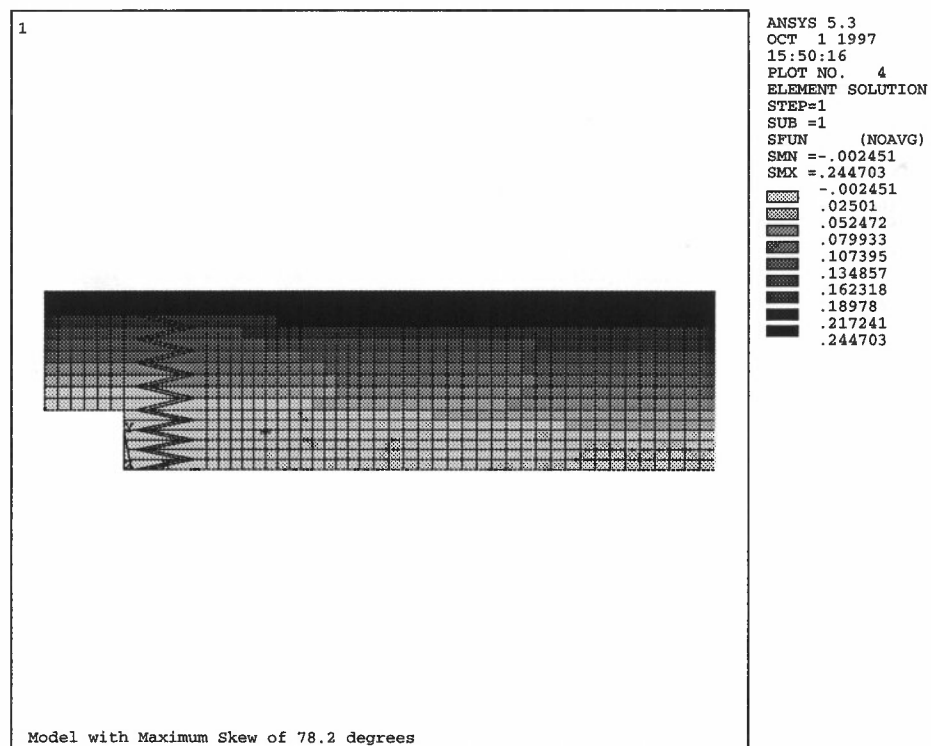


Figure 5.69: Contour Plot of Stream Function Variation; Non Averaged

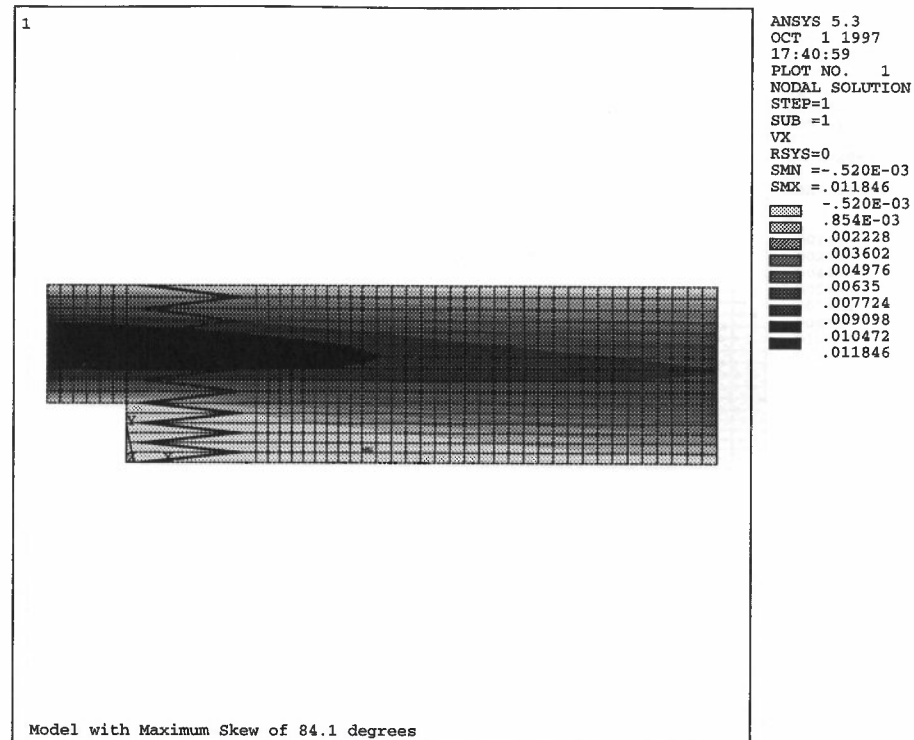


Figure 5.70: Contour Plot of Axial Velocity Variation

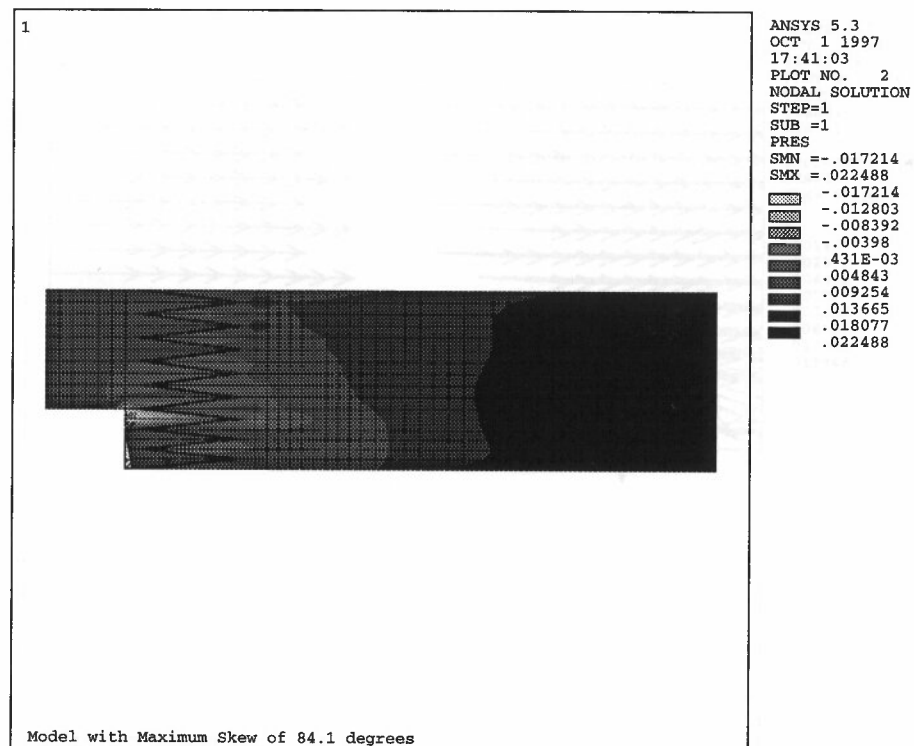


Figure 5.71: Contour Plot of Pressure Variation

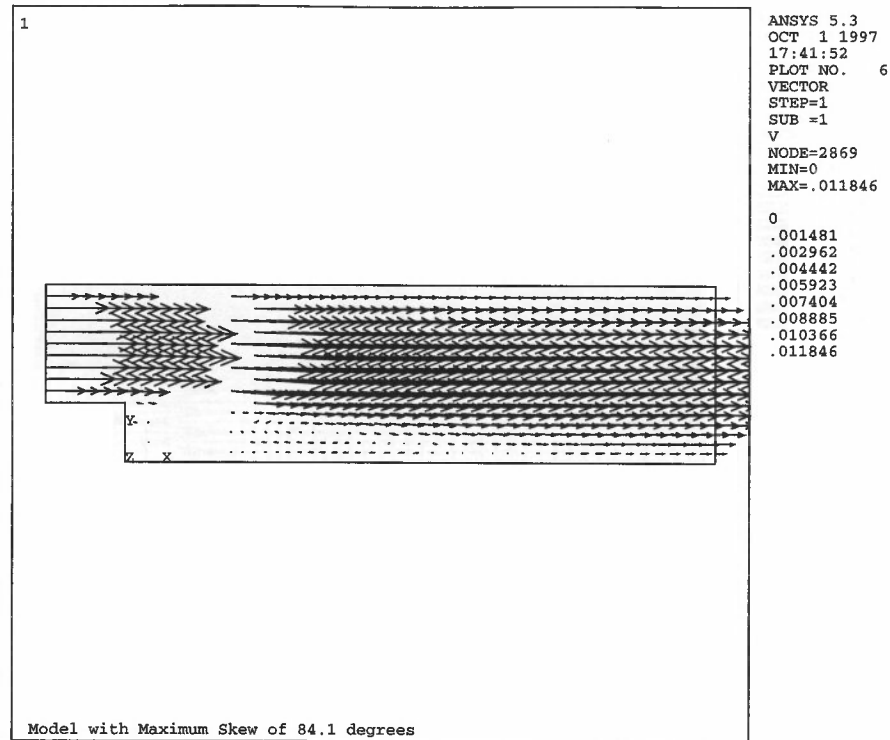


Figure 5.72: Vector Plot of Velocity Variation

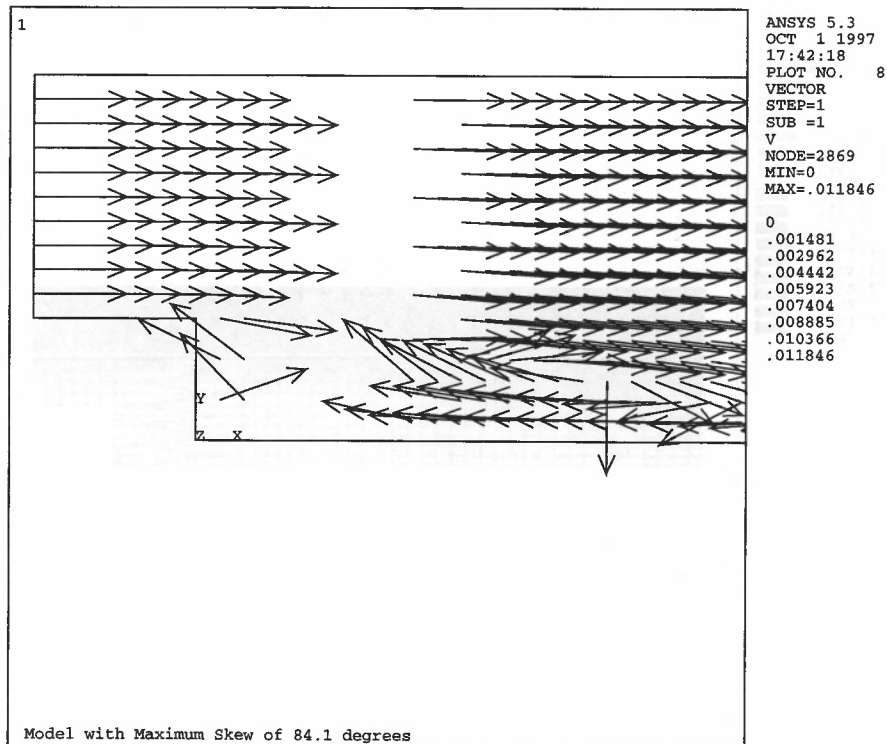


Figure 5.73: Magnification at Step location

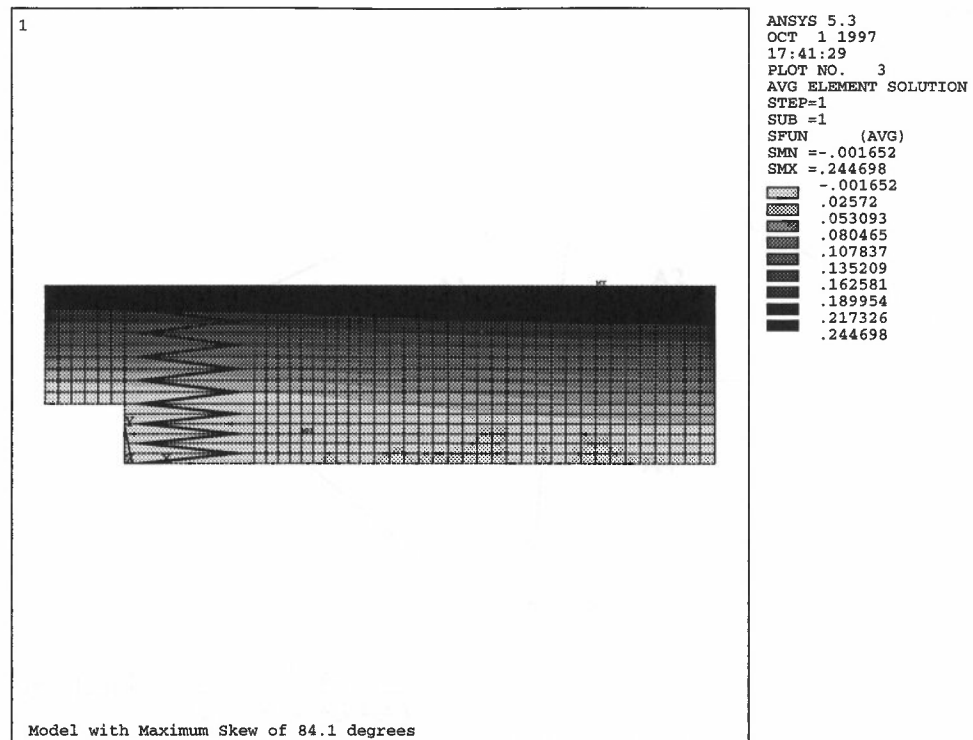


Figure 5.74: Contour Plot of Stream Function Variation; Averaged

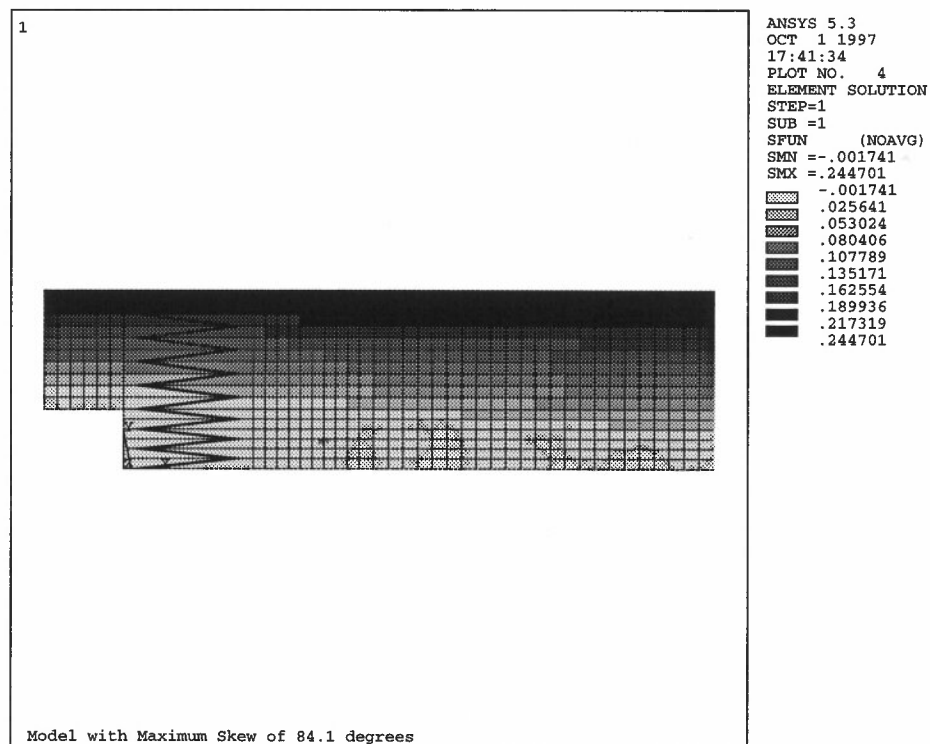


Figure 5.75: Contour Plot of Stream Function Variation; Non Averaged

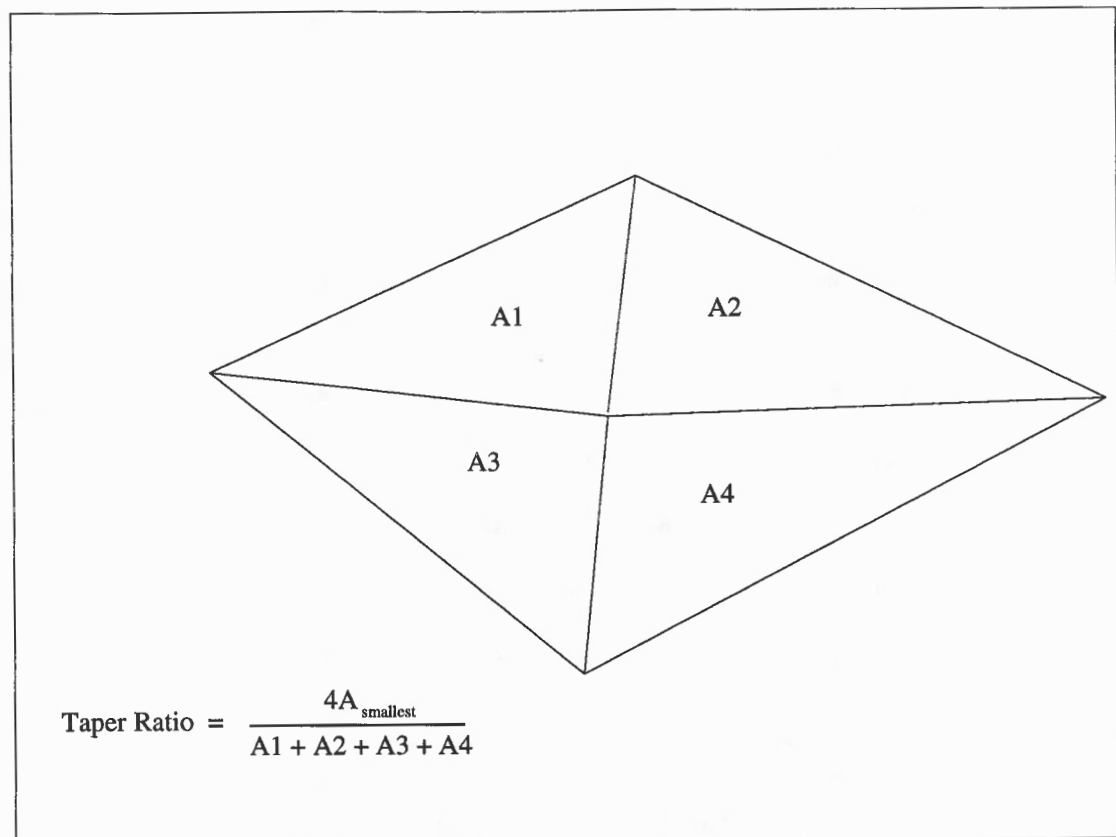


Figure 5.76: Quad Taper Ratio Definition

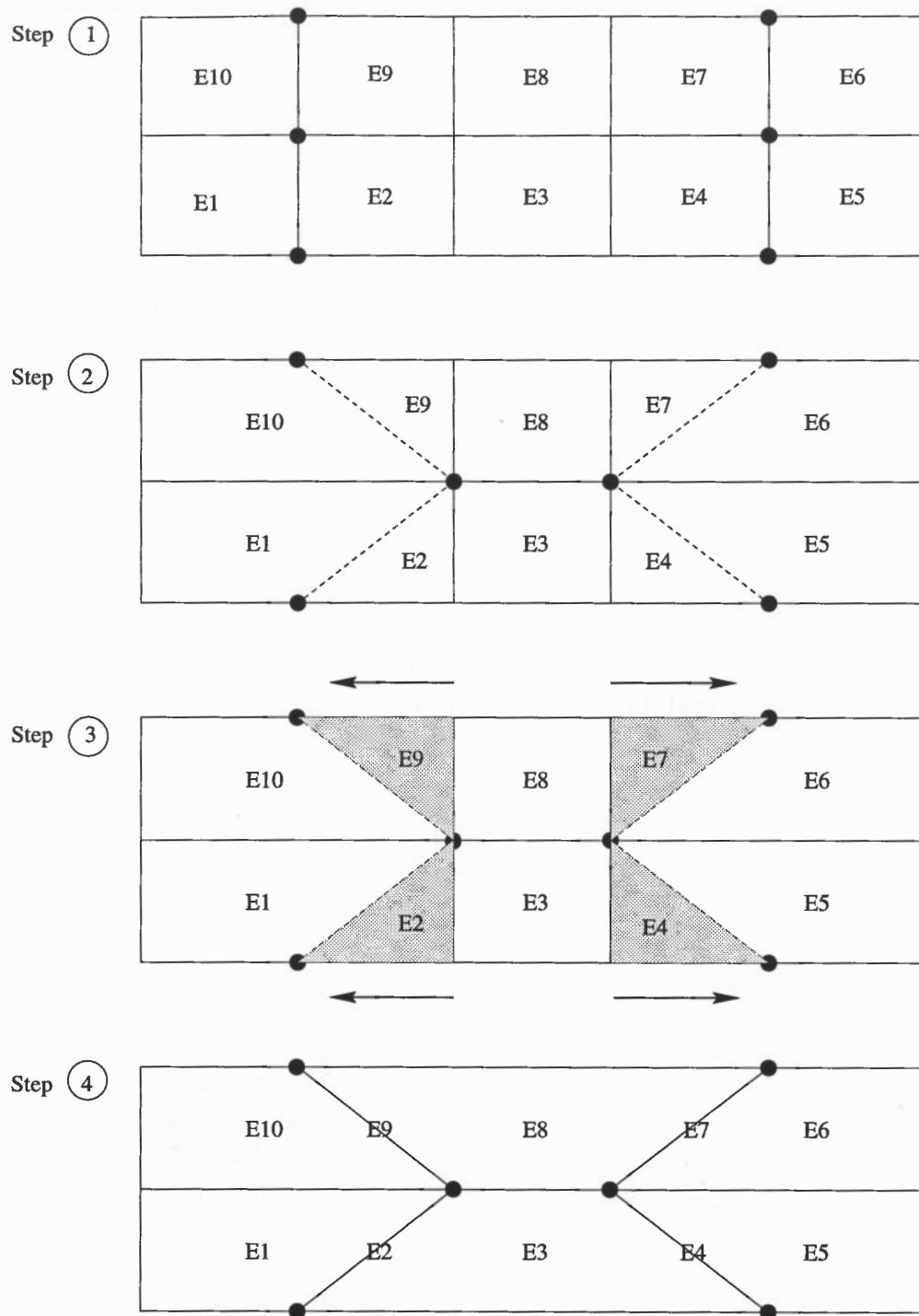


Figure 5.77: Illustration of Element Taper Methodology

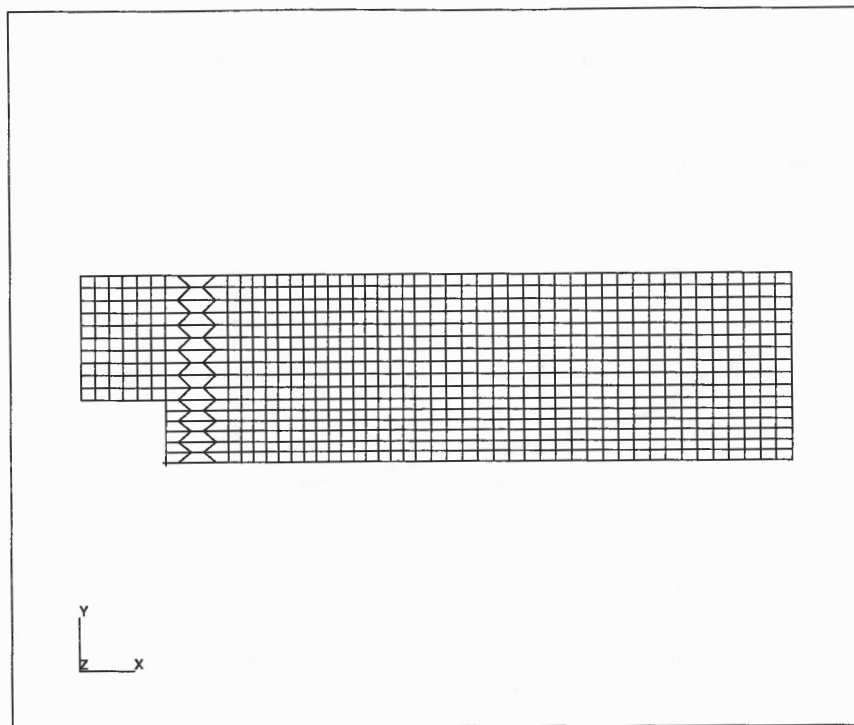


Figure 5.78: Mesh With Element Taper=0.5

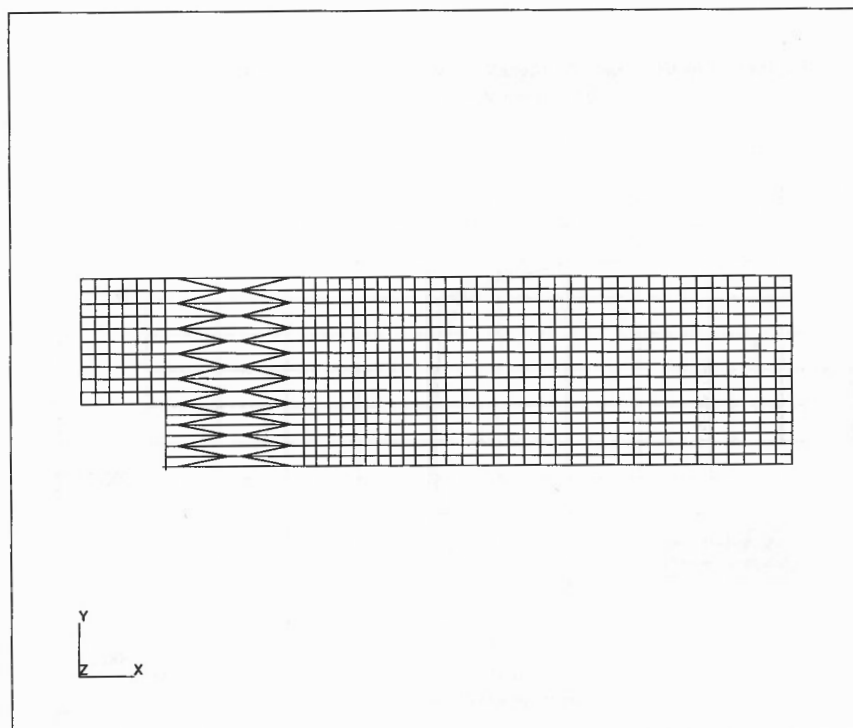


Figure 5.79: Mesh With Element Taper=0.2

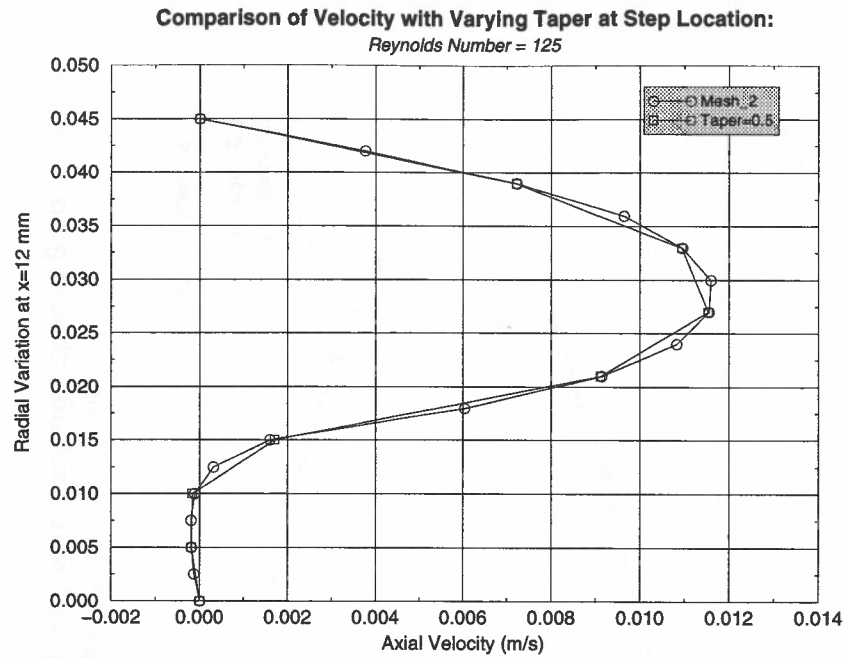


Figure 5.80: Taper Ratio Variation to Inspection Point One

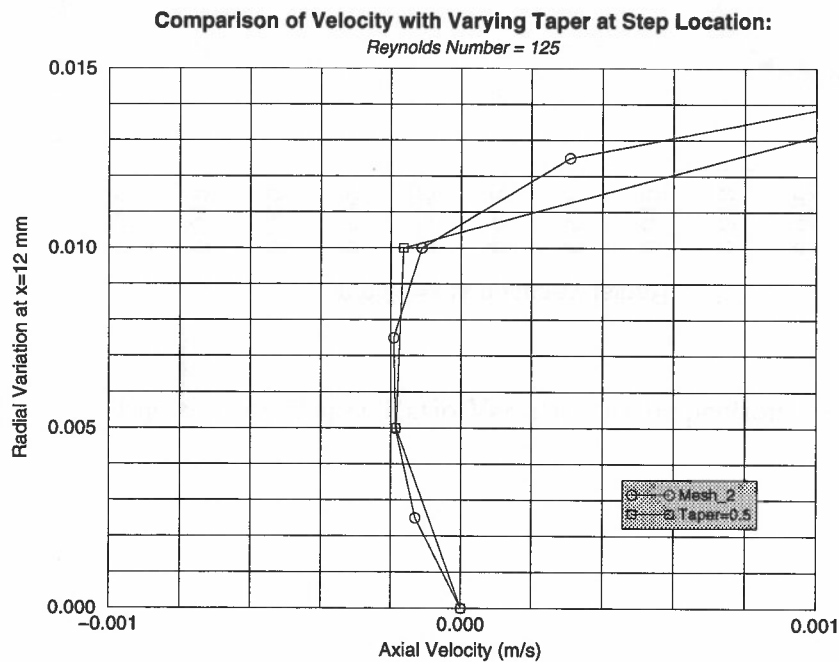


Figure 5.81: Magnification Plot at Step Location

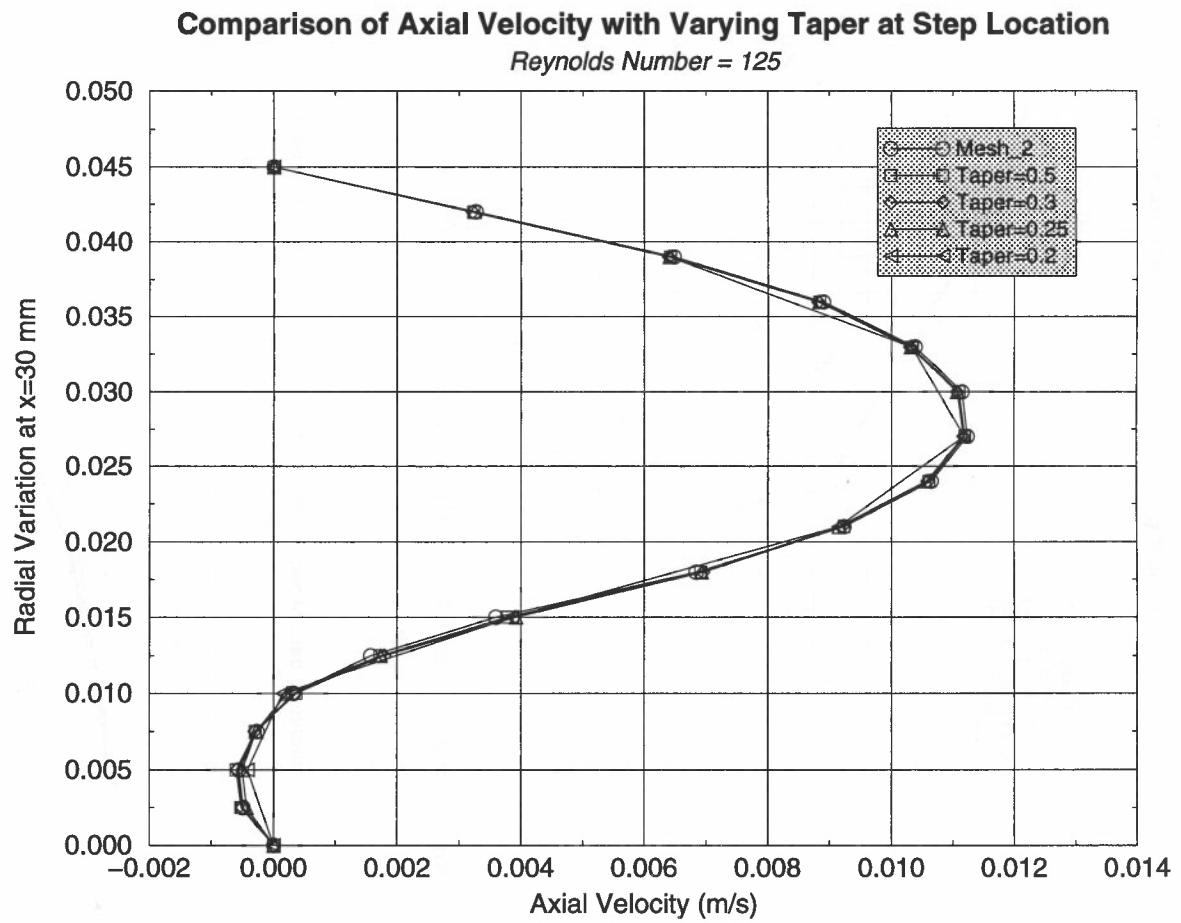


Figure 5.82: Taper Ratio Variation to Inspection Point Two

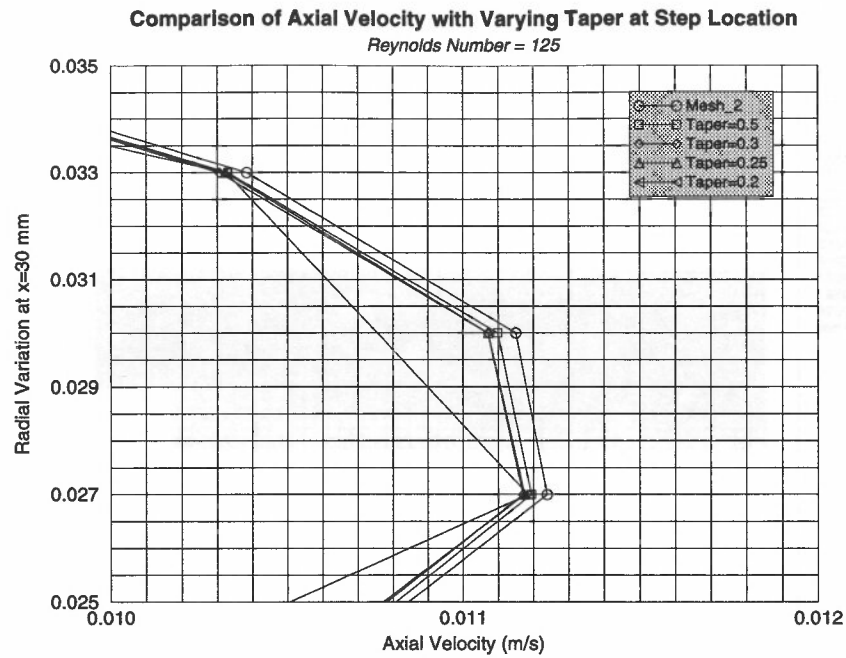


Figure 5.83: Magnification Plot at Channel Centre-line

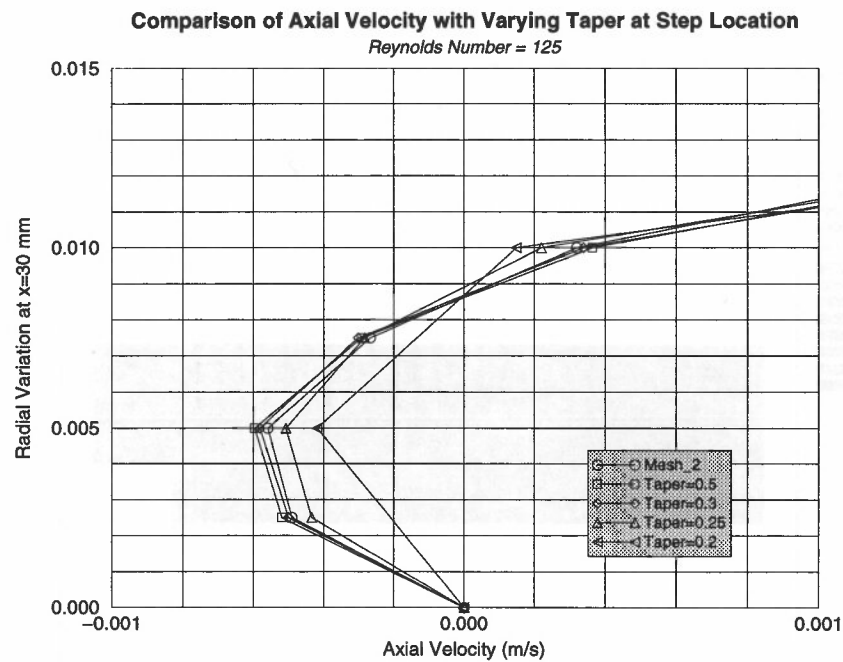


Figure 5.84: Magnification Plot at Step Location

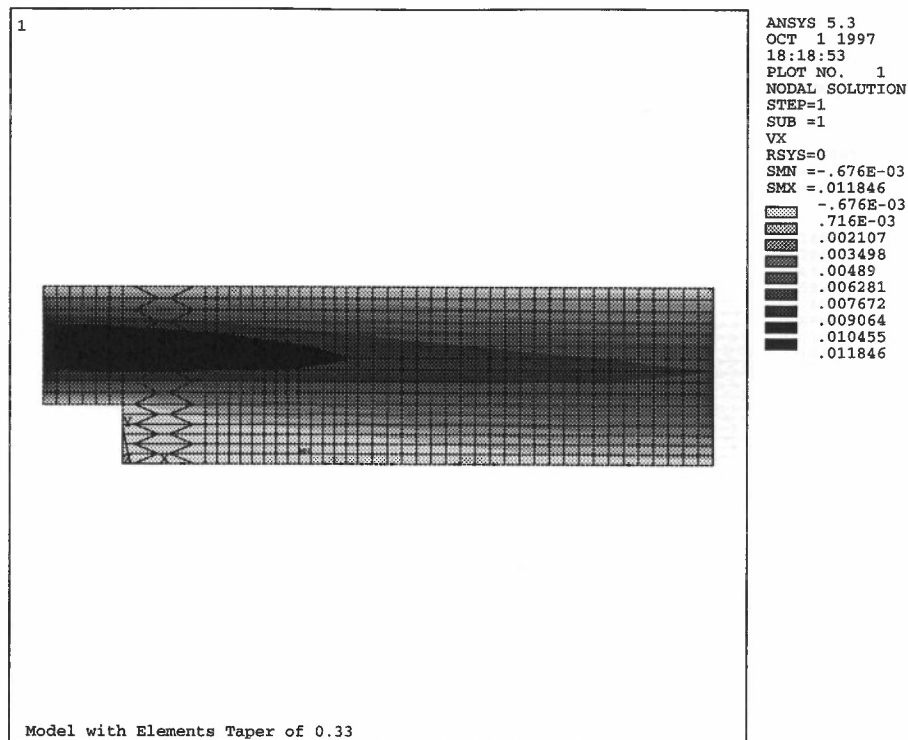


Figure 5.85: Contour Plot of Axial Velocity Variation

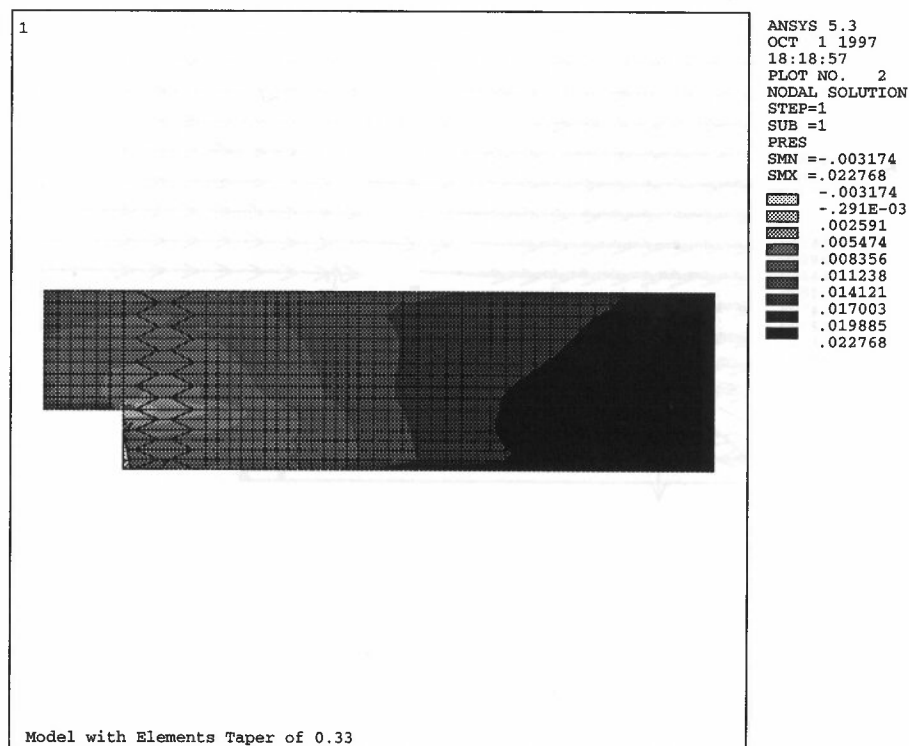


Figure 5.86: Contour Plot of Pressure Variation

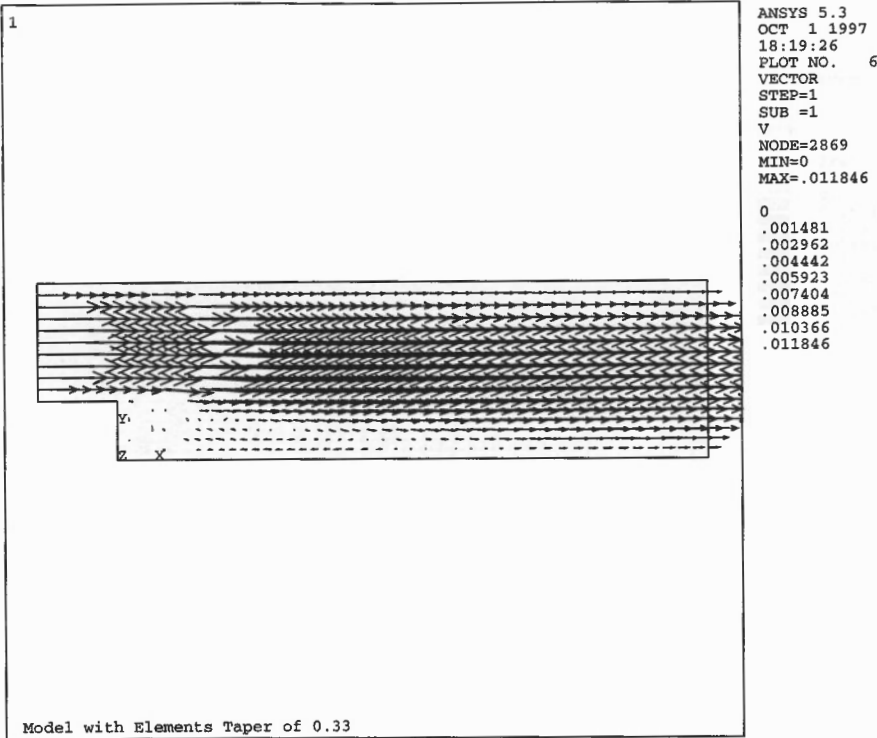


Figure 5.87: Vector Plot of Velocity Variation

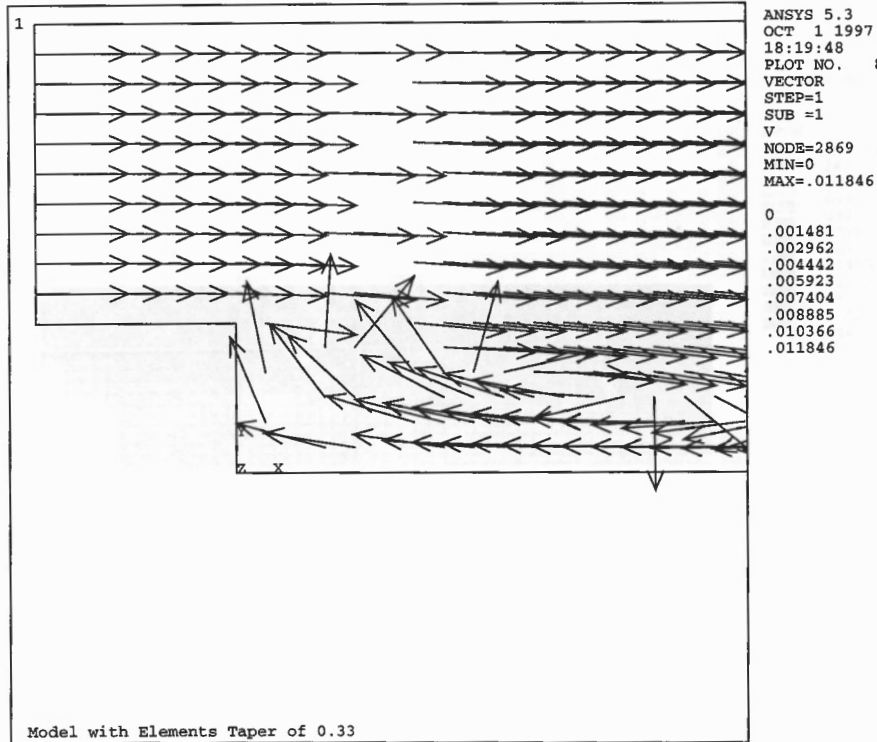


Figure 5.88: Magnification at Step location

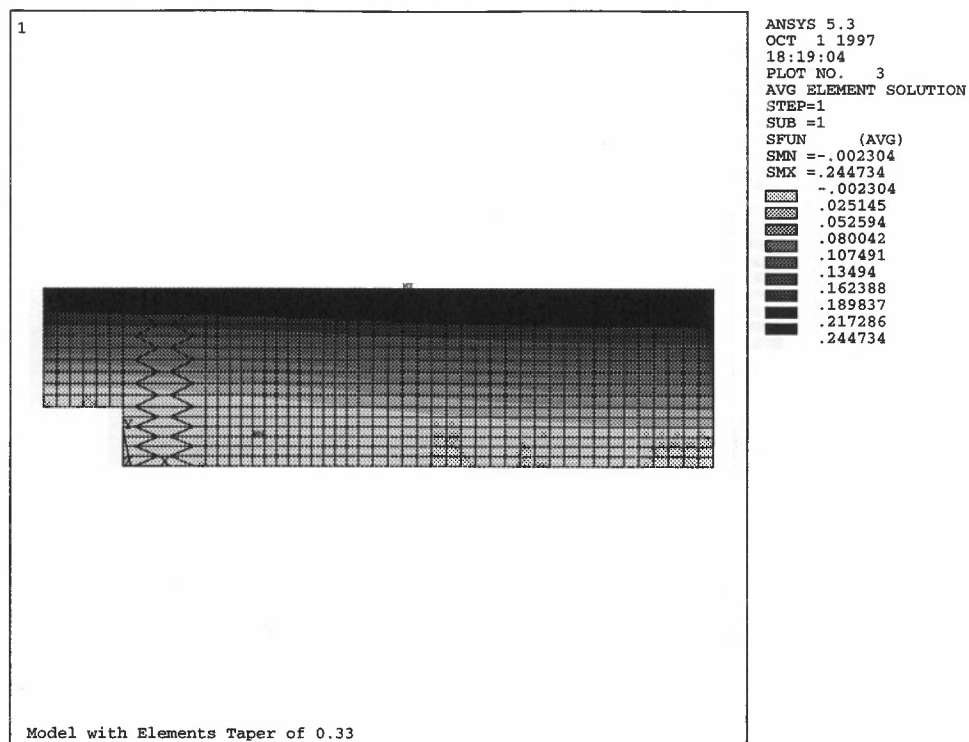


Figure 5.89: Contour Plot of Stream Function Variation; Averaged

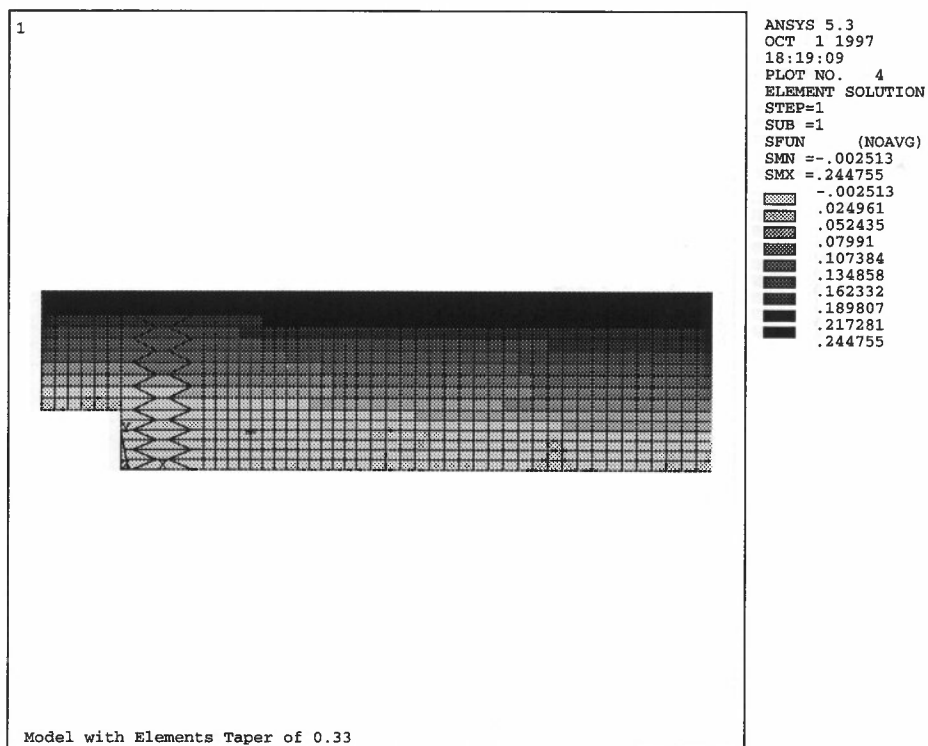


Figure 5.90: Contour Plot of Stream Function Variation; Non Averaged

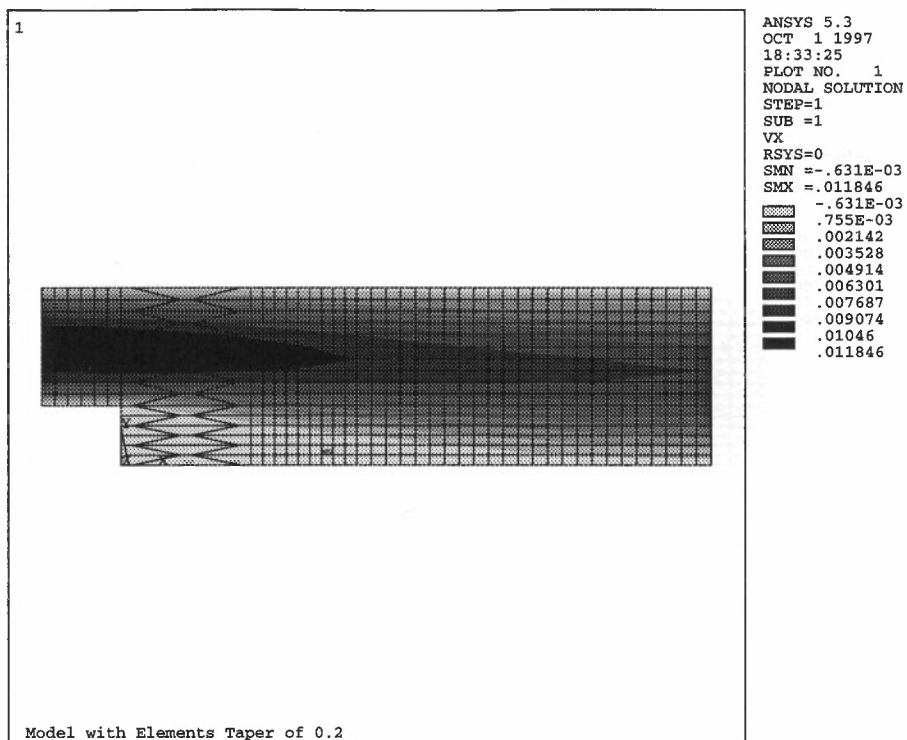


Figure 5.91: Contour Plot of Axial Velocity Variation

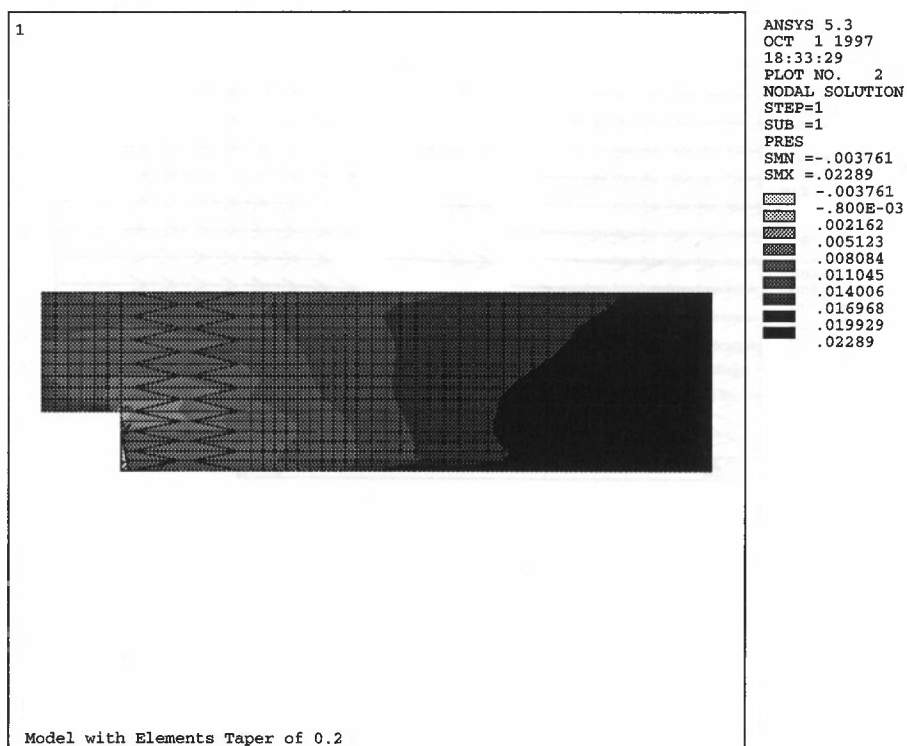


Figure 5.92: Contour Plot of Pressure Variation

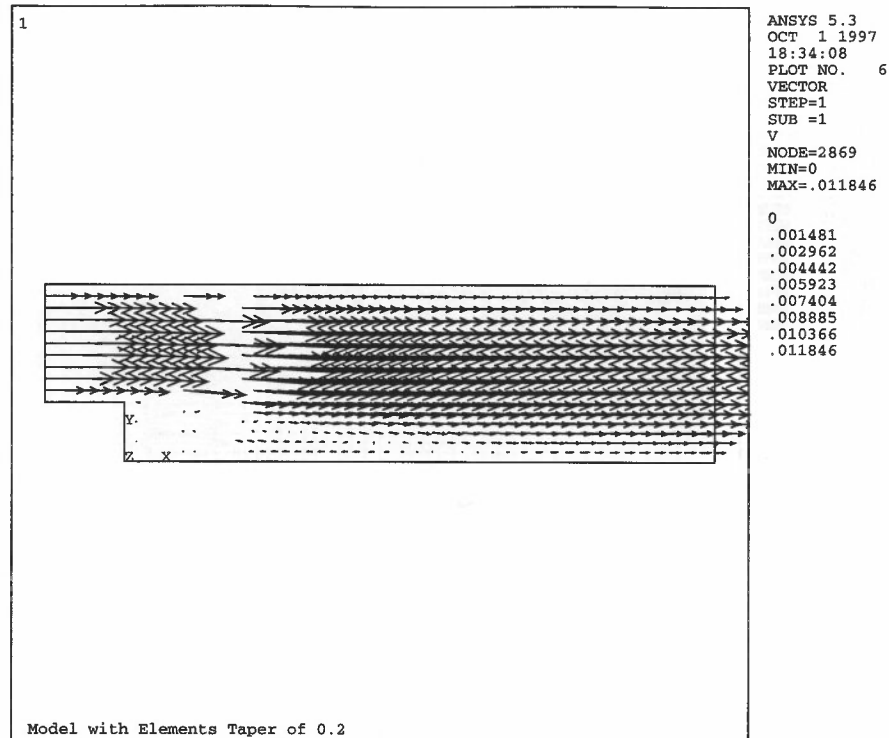


Figure 5.93: Vector Plot of Velocity Variation

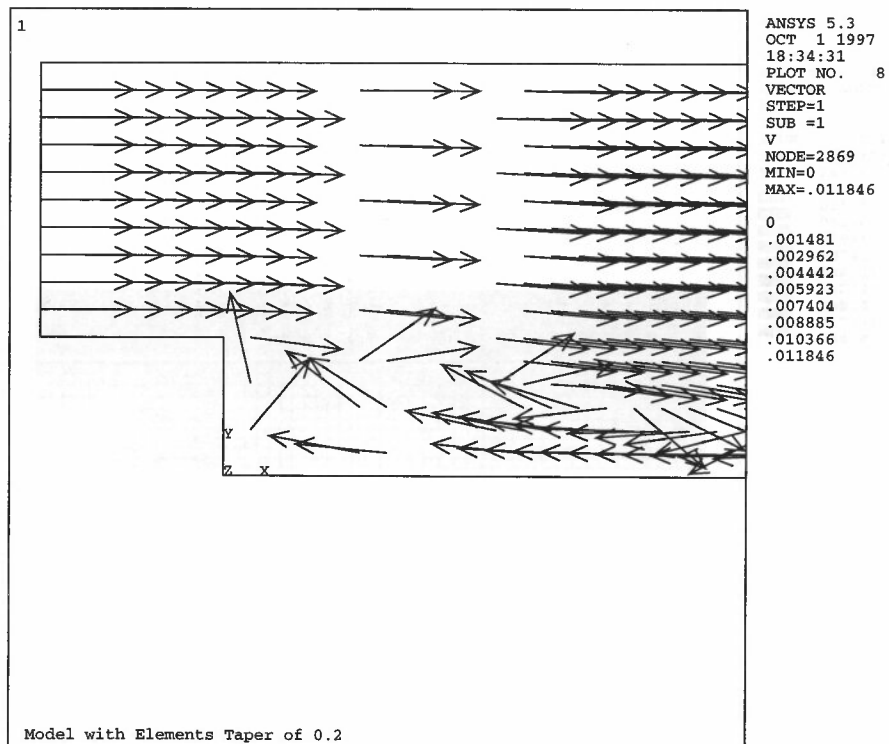


Figure 5.94: Magnification at Step location

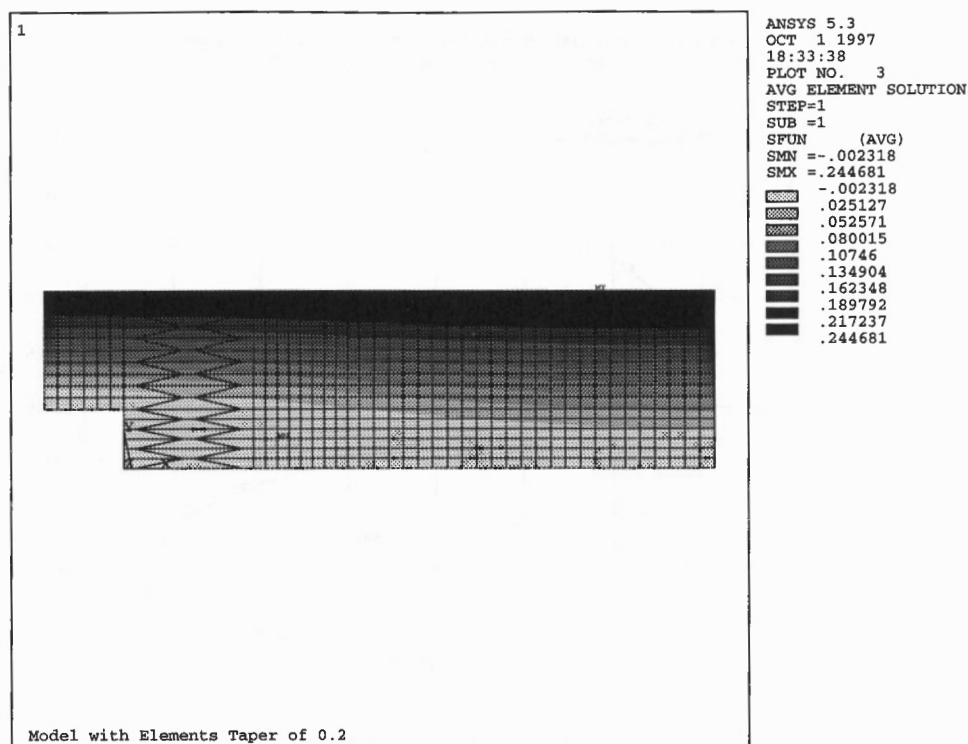


Figure 5.95: Contour Plot of Stream Function Variation; Averaged

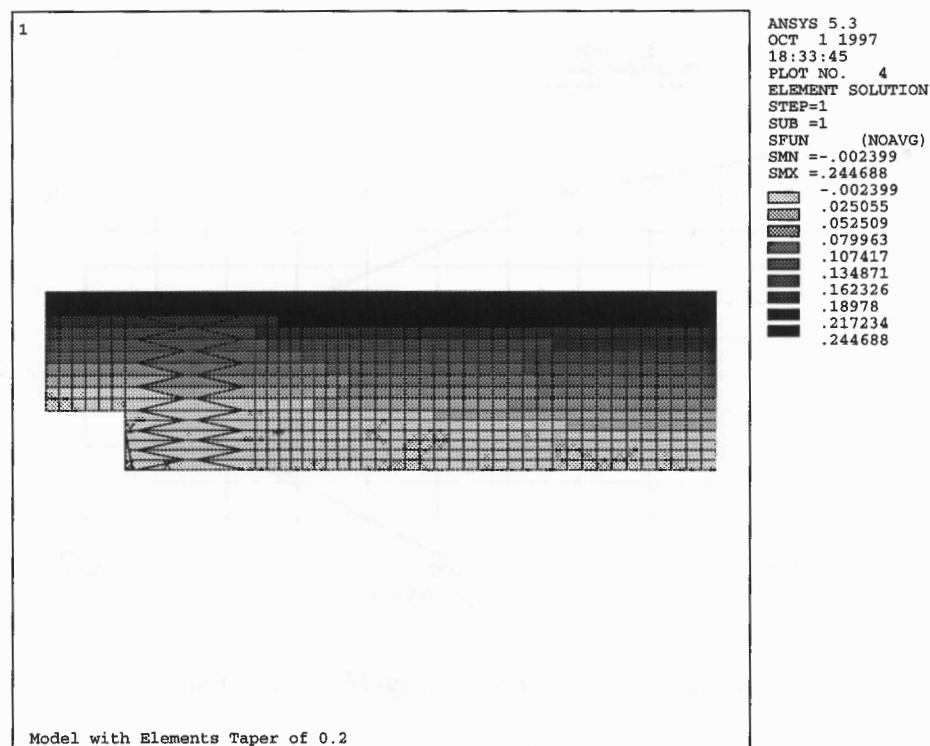


Figure 5.96: Contour Plot of Stream Function Variation; Non Averaged

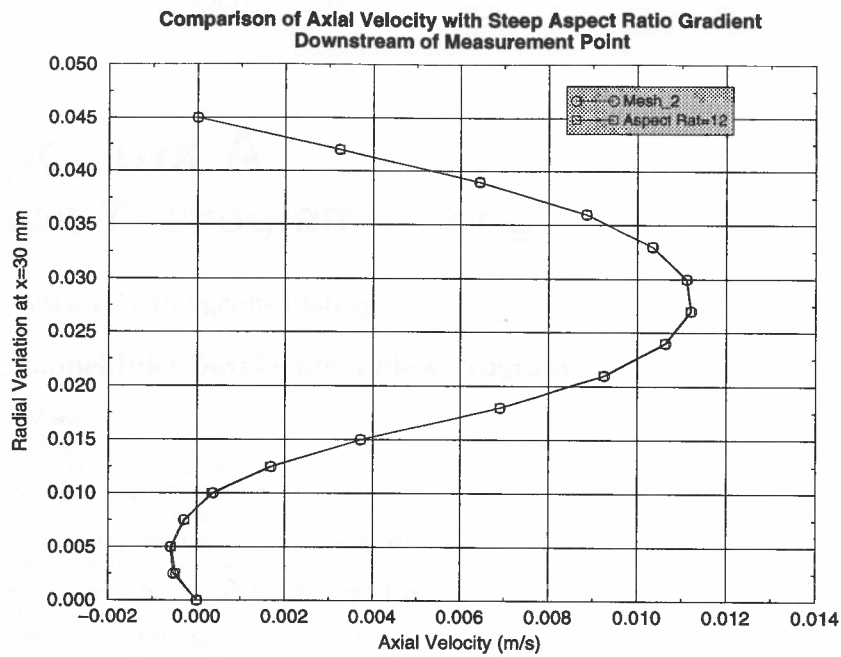


Figure 5.97: Perturbed Element Downstream of Inspection Point Two

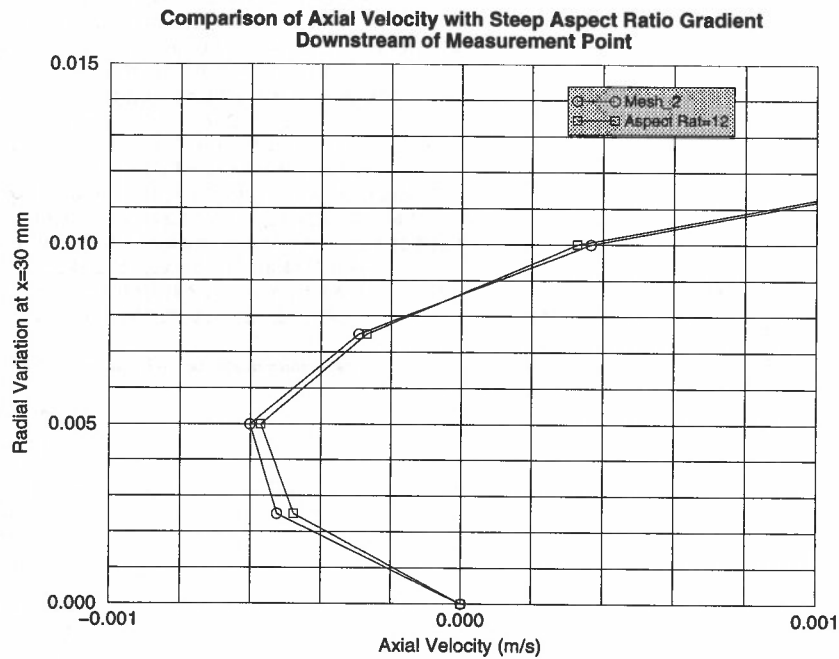


Figure 5.98: Magnification Plot at Step Location

Appendix A

Fortran-77 Program Listing

A.1.0 Fortran-77 Program Listing :

A.1.1 Channel Inlet Development Flow Program :

```

PROGRAM ans_devflow
IMPLICIT none
INTEGER dim,n_73,n_125,n_191
PARAMETER (dim=2000,n_73=21,n_191=20)
INTEGER i,node,ncount,icount
REAL x,y,dum1,dum2,dum3,dum4,x_prof,y_loc,x_vel
REAL y_pos,x_out,dum5,dum6,vel_fac,y_fac
REAL re73_prof,y73_loc,re125_prof,y125_loc,re191_prof
REAL y191_loc
DIMENSION x(dim),y(dim),node(dim),x_out(dim)
DIMENSION x_prof(21),y_loc(21),x_vel(21),y_pos(21)
DIMENSION re73_prof(n_73),y73_loc(n_73),re125_prof(n_73)
DIMENSION y125_loc(n_73),re191_prof(n_191),y191_loc(n_191)
CHARACTER *32 dum,ans
DATA x_prof/0.,0.30313,0.9094,1.9703,2.8797,4.244,5.4563,
& 6.1383,6.8203,7.1992,7.275,7.1992,6.669,6.0625,
& 5.15313,3.94063,2.425,1.6672,0.7578,0.2425,0./
DATA y_loc/15.0,15.25,15.5,16.875,18.125,20.0,21.875,23.75,
& 26.25,28.125,30.0,31.875,33.75,35.625,37.5,
& 39.375,41.875,42.5,43.75,44.375,45.0/
DATA re73_prof/0.,1.,3.,6.5,9.5,14.,18.,20.25,22.5,23.75,24.,
& 23.75,22.,20.,17.,13.,8.,5.5,2.5,0.8,0./
DATA y73_loc/0.,0.2,0.4,1.5,2.5,4.,5.5,7.0,9.,10.5,12.,13.5,
& 15.,16.5,18.,19.5,21.5,22.,23.,23.5,24./
DATA re125_prof/0.,2.,4.,7.3,10.,15.2,18.5,20.6,22.,22.8,23.,
& 22.5,21.5,20.,17.5,13.5,8.5,5.5,2.5,1.,0./
DATA y125_loc/0.,0.3,1.,1.5,2.5,4.,5.5,7.,8.75,10.25,11.75,
& 13.5,15.,16.8,18.2,20.,21.5,22.3,23.,23.3,24./
DATA re191_prof/0.,2.5,4.6,8.,11.4,16.3,19.5,21.3,22.2,22.5,
& 22.4,22.,21.,19.5,17.,13.5,8.4,5.5,1.6,0./
DATA y191_loc/0.,0.5,1.,1.5,2.5,4.,5.6,7.5,8.6,10.5,12.,13.5,
& 15.2,16.7,18.5,20.,21.5,22.3,23.2,24./
c
open(unit=11, file='devflw.nod', status='unknown')
c
vel_fac = 16.0
y_fac = 24.0
n_125 = n_73
c ----- Initialise Arrays -----
do i=1,dim
node(i) = 0
x(i) = 0.0
y(i) = 0.0
x_vel(i) = 0.0
x_out(i) = 0.0
y_pos(i) = 0.0
enddo
ncount = 1
10 continue
read(11,*,end=20) dum
if (dum(4:7) .eq. 'NODE') then
do i=1,20000
read(11,*,end=20) node(ncount),x(ncount),y(ncount),
& dum1,dum2,dum3,dum4
c write(6,*) node(ncount),x(ncount),y(ncount)
ncount = ncount + 1

```

```

        enddo
    else
        goto 10
    endif
20 continue
    icount = ncount - 1
    close(11)
    open(unit=12, file='devflw.out',status='unknown')
    write(6,9100)
    write(12,9100)
    do i=1,n_73
        x_vel(i) = 4.85*re73_prof(i)/vel_fac*0.001
        y_pos(i) = (15. + 30.*y73_loc(i)/y_fac)*0.001
c      write(6,*) y_pos(i)
    enddo
    call curfit(y_pos,x_vel,n_73,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
    do i=1,ncount-1
        write(6,*) x_out(i),y(i)
        write(12,9200) node(i),x_out(i)
    enddo
    write(6,9300)
    write(12,9300)
    do i=1,n_125
        x_vel(i) = 8.25*re125_prof(i)/vel_fac*0.001
        y_pos(i) = (15. + 30.*y125_loc(i)/y_fac)*0.001
    enddo
    call curfit(y_pos,x_vel,n_125,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
    do i=1,ncount-1
        write(6,*) x_out(i),y(i)
        write(12,9200) node(i),x_out(i)
    enddo
    write(12,9400)
    write(6,9400)
    do i=1,n_191
        x_vel(i) = 12.7*re191_prof(i)/vel_fac*0.001
        y_pos(i) = (15. + 30.*y191_loc(i)/y_fac)*0.001
    enddo
    call curfit(y_pos,x_vel,n_191,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
    write(6,*)
    do i=1,ncount-1
        write(6,*) x_out(i),y(i)
        write(12,9200) node(i),x_out(i)
    enddo
    close(12)
c
9000 format(a32)
9100 format('/COM Inlet Velocity Profile for Reynolds Number=73'
&      ',!')
9200 format('D',i7,',VX',e16.7)
9300 format('/COM Inlet Velocity Profile for Reynolds Number=125'
&      ',!')
9400 format('/COM Inlet Velocity Profile for Reynolds Number=191'
&      ',!')
c
END

```

A.1.2 Curve Fitting Routine Program :

```

C=====
C  QUADRATIC,CUBIC AND CUBIC SPLINE CURVE FITTING ROUTINES
C=====

```

```

C
C SUBROUTINE CURFIT(X,Y,N,X1,Y1,N1,ANS1,ANS2,GR1,GRN,ISP,LOP1,LOP2,
C & LOP3,NERR)
C-----
C
C VARIABLES -
C X - INPUT X ARRAY
C Y - INPUT Y ARRAY
C N - NUMBER OF (X,Y) POINTS (MAX=120 FOR CUBIC SPLINE FIT)
C
C X1 - INPUT X VALUES FOR WHICH INTERPOLATED Y
C VALUES REQUIRED (ISP=4)
C Y1 - OUTPUT Y1 VALUES CORRESPONDING TO X1 ARRAY
C N1 - NUMBER OF (X1,Y1) POINTS
C
C ANS1 - 'FIRST' ANSWER ARRAY
C = DY/DX AT (X,Y) FOR ISP=1 (1-N)
C = SLOPE IN RADIANs AT (X,Y) FOR ISP=2 (1-N)
C = INTEGRAL OF Y-DX FOR ISP=3 (1-N)
C
C ANS2 - 'SECOND' ANSWER ARRAY
C = D(DY/DX)/DX AT (X,Y) FOR ISP=1 (1-N)
C = CURVATURE AT (X,Y) FOR ISP=2 (1-N)
C
C GR1 - GRADIENT AT X=1 BOUNDARY CONDITION FOR CUBIC SPLINE (LOP=3)
C GRN - GRADIENT AT X=N BOUNDARY CONDITION FOR CUBIC SPLINE (LOP=3)
C
C ISP - CONTROL ON CURVE FITTING REQUIREMENT
C = 1 DYDX AND D(DY/DX)/DX CALCULATED
C = 2 SLOPE AND CURVATURE CALCULATED
C = 3 INTEGRAL FROM X=1 TO X=T CALCULATED
C = 4 INTERPOLATED Y1 VALUES CALCULATED
C
C LOP1 - TYPE OF CURVE FITTING REQUIRED
C = 1 LINEAR FIT
C = 2 QUADRATIC CURVE FIT
C = 3 CUBIC CURVE FIT
C = 4 CUBIC SPLINE FIT
C
C LOP2 - CURVE FIT OPTIONS
C = 0 STANDARD FITS
C = 1 AVERAGE OF CENTRAL AND END QUADRATICS FOR LOP1=2
C (ONLY AVAILABLE FOR GRADIENT OPTIONS - ISP=1 AND 2)
C END SECOND DERIVATIVE HALF PREVIOUS VALUE FOR LOP1=4
C
C LOP3 - EXTRAPOLATION OPTIONS
C LOP1 - TYPE OF CURVE FITTING REQUIRED
C = 0 STANDARD EXTRAPOLATION
C = 1 LINEAR EXTRAPOLATION FROM SLOPE AT END POINT
C
C NERR - OUTPUT STREAM FOR ERROR MESSAGES
C-----
C
C DIMENSION X(N),Y(N),X1(N1),Y1(N1),ANS1(N),ANS2(N),
C & A(150),B(150),C(150),D(150)
C-----
C----- INITIAL CALCULATIONS -----
C-----
C
C IF (ISP.EQ.3) ANS1(1) = 0.
C DO 20 K=1,N
C A(K) = 0.
C B(K) = 0.
C C(K) = 0.
C 20 D(K) = 0.
C
C-----
C----- CALCULATION OF REQUIRED CURVE FITTING TECHNIQUE -----

```

```

C-----
  IF (N.EQ.1) GOTO 70
  GOTO (30,40,50,60),LOP1
30 CALL LINEAR(X,Y,N,X1,Y1,N1,ANS1,ANS2,ISP,C,D)
  GOTO 80
40 IF (N.EQ.2) CALL LINEAR(X,Y,N,X1,Y1,N1,ANS1,ANS2,ISP,C,D)
  IF (N.GT.2) CALL QUAD(X,Y,N,X1,Y1,N1,ANS1,ANS2,ISP,LOP2,LOP3,
    &      B,C,D,NERR)
  GOTO 80
50 IF (N.EQ.2) CALL LINEAR(X,Y,N,X1,Y1,N1,ANS1,ANS2,ISP,C,D)
  IF (N.EQ.3) CALL QUAD(X,Y,N,X1,Y1,N1,ANS1,ANS2,ISP,LOP2,LOP3,
    &      B,C,D,NERR)
  IF (N.GT.3) CALL CUBE(X,Y,N,X1,Y1,N1,ANS1,ANS2,ISP,LOP2,LOP3,
    &      A,B,C,D)
  GOTO 80
60 CALL SPLINE(X,Y,N,X1,Y1,N1,ANS1,ANS2,GR1,GRN,ISP,LOP2,LOP3)
  GOTO 80
C
C-----
C INTERPOLATED VALUE SET FOR SINGLE POINT INPUT
C-----
C
70 IF (ISP.NE.4) GOTO 90
  DO 100 I=1,N1
100 Y1(I) = Y(1)
  RETURN
C
90 WRITE(NERR,600)
600 FORMAT(/' ***** ERROR IN CURFIT *****/'
  &' ***** OPTION NOT AVAILABLE FOR SINGLE POINT INPUT *****')
  STOP
80 IF (ISP.NE.2) RETURN
C
C-----
C SLOPE AND CURVATURE CALCULATED FROM FIRST AND SECOND DERIVATIVES
C-----
C
  DO 10 K=1,N
  ANS2(K) = ANS2(K)/(1. + ANS1(K)**2)**1.5
10 ANS1(K) = ATAN(ANS1(K))
  RETURN
  END

```



```

C=====
C      LINEAR FIT ROUTINE
C=====
C
  SUBROUTINE LINEAR(X,Y,N,X1,Y1,N1,ANS1,ANS2,ISP,C,D)
  DIMENSION X(1),Y(1),X1(1),Y1(1),ANS1(1),ANS2(1),C(1),D(1),
    &      BB(120),CC(120)
C
C----- LINEAR COEFFICIENTS CALCULATED (Y=C*X+D) -----
C-----
C
  NM1 = N-1
  DO 310 K=1,NM1
  C(K) = (Y(K+1) - Y(K))/(X(K+1) - X(K))
310 D(K) = Y(K) - C(K)*X(K)
  GOTO (300,300,360,370),ISP
C
C CALCULATION OF FIRST AND SECOND DERIVATIVES
C SECOND DERIVATIVE FROM AVERAGE OF TWO QUADRATICS ASSUMING
C THE SLOPE FROM THE LINEAR OPTION AT THE GIVEN POINT

```

```

C-----
C
300  ANS1(1) = C(1)
      ANS1(N) = C(NM1)
      DO 320 K=2,NM1
320  ANS1(K) = .5*(C(K-1) + C(K))
      DO 330 K=1,NM1
330  BB(K) = (ANS1(K)*(X(K) - X(K+1)) - (Y(K) - Y(K+1)))/(X(K) -
      &      X(K+1))**2
      DO 340 K=2,N
340  CC(K) = (ANS1(K)*(X(K) - X(K-1)) - (Y(K) - Y(K-1)))/(X(K) -
      X(K-1))**2
      ANS2(1) = 2.*BB(1)
      ANS2(N) = 2.*CC(N)
c
      DO 350 K=2,NM1
350  ANS2(K) = BB(K) + CC(K)
      RETURN
C-----
c----- INTEGRATION -----
C-----
C
360  DO 10 K=2,N
10  ANS1(K) = ANS1(K-1) + .5*(Y(K) + Y(K-1))*(X(K) - X(K-1))
      RETURN
c
C-----
C----- INTERPOLATION -----
C-----
C
370  DO 20 I=1,N1
      DO 30 K=2,N
      IF (X1(I).LT.X(K)) GO TO 40
30  CONTINUE
      K = N
40  Y1(I) = C(K-1)*X1(I) + D(K-1)
20  CONTINUE
      RETURN
      END
C
C=====
C      QUADRATIC CURVE FITTING ROUTINE
C=====
C
      SUBROUTINE QUAD(X,Y,N,X1,Y1,N1,ANS1,ANS2,ISP,LOP2,LOP3,B,C,D,NERR)
      DIMENSION X(1),Y(1),X1(1),Y1(1),ANS1(1),ANS2(1),B(1),C(1),D(1)
      NM2 = N-2
      NM1 = N-1
      NM3 = N-3
C-----
C      QUADRATIC COEFFICIENTS B,C AND D CALCULATED ( Y=B*X**2+C*X+D )
C-----
C
      CALL QUIN(X,Y,N,B,C,D)
      IF (LOP2.EQ.1) GOTO 200
C-----
C----- OPTION 1 -----
C-----
C
45  CONTINUE
      GOTO (20,20,20,30),ISP
C
C-----
C      CALCULATION OF FIRST AND SECOND DERIVATIVES OR INTEGRATION
C-----
C
20  DO 22 K=1,N

```



```

      IF (K.EQ.1) KK = 1
      IF (K.EQ.N) KK = NM2
      IF (K.NE.1.AND.K.NE.N) KK = K - 1
      GOTO (24,24,26,22),ISP
C
C -----
C  CALCULATION OF FIRST AND SECOND DERIVATIVES
C -----
C
24  ANS1(K) = 2.*B(KK)*X(K) + C(KK)
     ANS2(K) = 2.*B(KK)
     GOTO 22
C
C -----
C  CALCULATION OF INTEGRAL - AVERAGE OF THE TWO AVAILABLE QUADRATICS
C -----
C
26  IF (K.EQ.1) GOTO 22
     ANS1(K) = ANS1(K-1) + B(KK)*(X(K)**3 - X(K-1)**3)/3. + C(KK)*
     &(X(K)**2 - X(K-1)**2)/2. + D(KK)*(X(K) - X(K-1))
     IF (K.EQ.N.OR.K.EQ.2) GOTO 22
     KM = KK - 1
     DUM = ANS1(K-1) + B(KM)*(X(K)**3 - X(K-1)**3)/3. + C(KM)*
     &(X(K)**2 - X(K-1)**2)/2. + D(KM)*(X(K) - X(K-1))
     ANS1(K) = .5*(ANS1(K) + DUM)
22  CONTINUE
     RETURN
C
C -----
C  INTERPOLATION FROM AVERAGE OF THE TWO AVAILABLE QUADRATICS
C -----
C
30  DO 32 I=1,N1
     IF (X1(I).LT.X(1).AND.LOP3.EQ.1) GOTO 40
     IF (X1(I).GT.X(N).AND.LOP3.EQ.1) GOTO 50
     DO 34 J=3,N
     IF (X1(I).LT.X(J)) GOTO 33
34  CONTINUE
     J = N
33  JJ = J - 2
     Y1(I) = B(JJ)*X1(I)**2 + C(JJ)*X1(I) + D(JJ)
     DO 36 J=2,NM1
     IF (X1(I).LT.X(J)) GOTO 37
36  CONTINUE
     J = NM1
37  JJ = J - 1
     DUM = B(JJ)*X1(I)**2 + C(JJ)*X1(I) + D(JJ)
     Y1(I) = .5*(Y1(I) + DUM)
     GOTO 32
C
C -----
C  EXTRAPOLATION USING LINEAR FIT AT END POINT SLOPE
C -----
C
40  GRAD = 2.*B(1)*X(1) + C(1)
     Y1(I) = Y(1) - GRAD*(X(1) - X1(I))
     GOTO 32
50  GRAD = 2.*B(NM2)*X(N) + C(NM2)
     Y1(I) = Y(N) + GRAD*(X1(I) - X(N))
32  CONTINUE
     RETURN
C
C -----
C  OPTION 2 - FIRST AND SECOND DERIVATIVES FROM THE CENTRAL AND
C  THE TWO END DERIVATIVES
C  DOUBLE WEIGHT ON CENTRAL DERIVATIVE
C -----
C

```

```

200 GOTO (110,110,130,140) ISP
110 IF (N.LT.5) GOTO 210
    DO 220 K=3,NM2
        K1 = K - 1
        K2 = K - 2
        ANS1(K) = .25*(X(K)*(4.*B(K1) + 2.*B(K2) + 2.*B(K)) + (2.*C(K1) +
&      C(K2) + C(K)))
        ANS2(K) = .25*(2.*B(K2) + 4.*B(K1) + 2.*B(K))
220 CONTINUE
210 ANS1(1) = 2.*B(1)*X(1) + C(1)
    ANS1(2) = .3333*(X(2)*(4.*B(1) + 2.*B(2)) + (2.*C(1) + C(2)))
    ANS1(NM1) = .3333*(X(NM1)*(4.*B(NM2) + 2.*B(NM3)) +
&      (2.*C(NM2) + C(NM3)))
    ANS1(N) = 2.*B(NM2)*X(N) + C(NM2)
    ANS2(1) = 2.*B(1)
    ANS2(2) = .3333*(4.*B(1) + 2.*B(2))
    ANS2(NM1) = .3333*(4.*B(NM2) + 2.*B(NM3))
    ANS2(N) = 2.*B(NM2)
    RETURN
130 CONTINUE
140 CONTINUE
    WRITE(NERR,600)
600 FORMAT(/' ***** WARNING - WEIGHTED OPTION NOT AVAILABLE ***** ',
&' ***** USING STANDARD QUADRATIC FIT *****')
    GOTO 45
END

```

C

```

=====
C ROUTINE TO CALCULATE QUADRATIC COEFFICIENTS
=====
C

```

```

SUBROUTINE QUIN(X,Y,N,A,B,C)
DIMENSION X(1),Y(1),A(1),B(1),C(1)
NM2 = N - 2
DO 20 K=1,NM2
    DUM = (X(K+2)**2 - X(K)**2)*(X(K+2) - X(K+1)) - (X(K+2)**2 -
&      X(K+1)**2)*(X(K+2) - X(K))
    A(K) = ((Y(K+2) - Y(K))*(X(K+2) - X(K+1)) - (Y(K+2) - Y(K+1))*
&      (X(K+2) - X(K)))/DUM
    B(K) = ((Y(K+2) - Y(K+1)) - A(K)*(X(K+2)**2 - X(K+1)**2))/
&      (X(K+2) - X(K+1))
20 C(K) = Y(K) - A(K)*X(K)**2 - B(K)*X(K)
RETURN
END

```

```

=====
C ROUTINE TO INTERPOLATE FROM KNOWN QUADRATIC COEFFICIENTS
=====
C

```

```

SUBROUTINE QUOUT(X,B,C,D,N,X1,Y1)
DIMENSION X(1),B(1),C(1),D(1)
DO 10 J=3,N
    IF (X1.LT.X(J)) GOTO 20
10 CONTINUE
    J = N
20 J1 = J - 1
    J2 = J - 2
    Y1 = 0.
    DO 30 ID=1,2
        K = J2
        IF (X1.LT.X(2).AND.ID.EQ.2) GOTO 30
        IF (J1.EQ.N-1.AND.ID.EQ.2) GOTO 30
        IF (ID.EQ.2) K = J1
        Y1 = Y1 + B(K)*X1**2 + C(K)*X1 + D(K)
        IF (ID.EQ.2) Y1 = Y1/2.
30

```

```

30 CONTINUE
   RETURN
   END

```

```

C=====
C SPLINE CURVE FITTING SUBROUTINE
C=====
C

```

```

   SUBROUTINE SPLINE(X,Y,N,X1,Y1,N1,ANS1,ANS2,GR1,GRN,ISP,LOP2,LOP3)
   DIMENSION X(1),Y(1),S(120),A(120),B(120),C(120),F(120),
   &    W(120),SB(120),G(120),ANS1(1),ANS2(1),X1(1),Y1(1)
   DO 10 I=2,N
10  S(I) = X(I) - X(I-1)
      IF (N.LT.3) GOTO 130
      NO = N-1
      DO 20 I=2,NO
      A(I) = S(I)/6.
      B(I) = (S(I) + S(I+1))/3.
      C(I) = S(I+1)/6.
20  F(I) = (Y(I+1) - Y(I))/S(I+1) - (Y(I) - Y(I-1))/S(I)
130 CONTINUE
      IF (LOP2.EQ.0) GOTO 110

```

```

C
C-----
C BOUNDARY CONDITIONS FOR END POINT SECOND DERIVATIVES
C-----
C

```

```

   A(N) = -.5
   B(1) = 1.
   B(N) = 1.
   C(1) = -.5
   F(1) = 0.
   F(N) = 0.
   G(1) = 0.
   GO TO 120

```

```

C
C-----
C BOUNDARY CONDITIONS FOR END POINT GRADIENTS
C-----
C

```

```

110 A(N) = S(N)/6.
    B(1) = S(2)/3.
    B(N) = S(N)/3.
    C(1) = S(2)/6.
    F(1) = (Y(2) - Y(1))/S(2) - GR1
    F(N) = GRN - (Y(N) - Y(N-1))/S(N)
    G(1) = F(1)/B(1)

```

```

120 CONTINUE

```

```

C
C-----
C PIECEWISE CUBIC PARAMETERS CALCULATED
C-----
C

```

```

   W(1) = B(1)
   SB(1) = C(1)/W(1)
   DO 30 I=2,N
   W(I) = B(I) - A(I)*SB(I-1)
   SB(I) = C(I)/W(I)
30  G(I) = (F(I) - A(I)*G(I-1))/W(I)

```

```

C
C-----
C CALCULATION OF SECOND DERIVATIVES
C-----
C

```

```

   ANS2(N) = G(N)
   DO 40 I=2,N
   K = N + 1 - I

```

```

40  ANS2(K) = G(K) - SB(K)*ANS2(K+1)
    GOTO (100,100,200,300),ISP
C-----
C  CALCULATION OF FIRST DERIVATIVES
C-----
C
100 ANS1(1) = -S(2)/6.*(2.*ANS2(1) + ANS2(2)) + (Y(2)-Y(1))/S(2)
    DO 50 I=2,N
50  ANS1(I) = S(I)/6.*(2.*ANS2(I) + ANS2(I-1)) + (Y(I)-Y(I-1))/S(I)
    RETURN
C
C-----
C----- INTEGRATION -----
C-----
C
200 DO 60 I=2,N
60  ANS1(I) = ANS1(I-1) + S(I)*(Y(I)+Y(I-1))/2.-S(I)**3*
    &    (ANS2(I) + ANS2(I-1))/24.
    RETURN
C
C-----
C----- INTERPOLATION -----
C-----
C
300 DO 90 I=1,N1
    IF (X1(I).LT.X(1).AND.LOP3.EQ.1) GOTO 250
    IF (X1(I).GT.X(N).AND.LOP3.EQ.1) GOTO 350
    K = 2
    IF (X1(I)-X(1)) 85,54,70
54  Y1(I) = Y(1)
    GOTO 90
70  IF (X1(I)-X(K)) 85,75,80
75  Y1(I) = Y(K)
    GOTO 90
80  K = K+1
    IF (K-N) 70,85,85
85  Y1(I) = ANS2(K-1)*(X(K) - X1(I))**3/6./S(K) + ANS2(K)*
    &    (X1(I) - X(K-1))**3/6./S(K) + (Y(K)/S(K) -
    &    ANS2(K)*S(K)/6.)*(X1(I) - X(K-1)) + (Y(K-1)/S(K) -
    &    ANS2(K-1)*S(K)/6.)*(X(K) - X1(I))
C
C-----
C  EXTRAPOLATION USING LINEAR FIT AT END POINT SLOPE
C-----
C
    GOTO 90
250 GRAD = -S(2)/6.*(2.*ANS2(1)+ ANS2(2)) + (Y(2) - Y(1))/S(2)
    Y1(I) = Y(1) - GRAD*(X(1) - X1(I))
    GOTO 90
350 GRAD = S(N)/6.*(2.*ANS2(N) + ANS2(N-1)) + (Y(N) - Y(N-1))/S(N)
    Y1(I) = Y(N) + GRAD*(X1(I) - X(N))
90  CONTINUE
    RETURN
    END

C=====
C  ROUTINE FOR CUBIC CURVE FITTING
C=====
C
SUBROUTINE CUBE(X,Y,N,X1,Y1,N1,ANS1,ANS2,ISP,LOP2,LOP3,A,B,C,D)
DIMENSION X(1),Y(1),X1(1),Y1(1),ANS1(1),ANS2(1),A(1),B(1),C(1),
&    D(1)
NM3 = N-3
CALL CUBIN(X,Y,N,A,B,C,D)
GOTO (100,100,200,300),ISP
c
C-----

```

C CALCULATION OF FIRST AND SECOND DERIVATIVES

C

C

```

100 DO 110 K=1,N
    K1 = K-2
    K2 = K-1
    IF (K1.LT.1) K1 = 1
    IF (K2.LT.1) K2 = 1
    IF (K1.GT.NM3) K1 = NM3
    IF (K2.GT.NM3) K2 = NM3
    ANS1(K) = 1.5*(A(K1) + A(K2))*X(K)**2 + (B(K1) + B(K2))*X(K) +
    &      0.5*(C(K1) + C(K2))
110 ANS2(K) = 3.0*(A(K1) + A(K2))*X(K) + B(K1) + B(K2)
    RETURN

```

C

C

C

C

C

```

200 DO 210 K=2,N
    K1 = K-2
    IF (K1.LT.1) K1 = 1
    IF (K1.GT.NM3) K1 = NM3
210 ANS1(K) = ANS1(K-1) + A(K1)*(X(K)**4 - X(K-1)**4)/4. +
    &      B(K1)*(X(K)**3 - X(K-1)**3)/3. + C(K1)*(X(K)**2 -
    &      X(K-1)**2)/2. + D(K1)*(X(K) - X(K-1))
    RETURN

```

C

C

C

C

C

```

300 DO 310 I=1,N1
    IF (X1(I).LT.X(1).AND.LOP3.EQ.1) GOTO 350
    IF (X1(I).GT.X(N).AND.LOP3.EQ.1) GOTO 360
c
    DO 320 K=1,N
    IF (X1(I).LT.X(K)) GO TO 330
320 CONTINUE
    K = N
330 K1 = K-2
    IF (K1.LT.1) K1 = 1
    IF (K1.GT.NM3) K1 = NM3
    Y1(I) = A(K1)*X1(I)**3 + B(K1)*X1(I)**2 + C(K1)*X1(I) + D(K1)
    GOTO 310

```

C

C

C

C

C

```

C EXTRAPOLATION USING LINEAR FIT AT END POINT SLOPE
C
C
350 GRAD = 3.*A(1)*X(1)**2 + 2.*B(1)*X(1) + C(1)
    Y1(I) = Y(1) - GRAD*(X(1) - X1(I))
    GOTO 310
c
360 GRAD = 3.*A(NM3)*X(N)**2 + 2.*B(NM3)*X(N) + C(NM3)
    Y1(I) = Y(N) + GRAD*(X1(I) - X(N))
c
310 CONTINUE
    RETURN
    END

```

C

C

C

C

C

C

C

C

C

C

ROUTINE TO CALCULATE CUBIC COEFFICIENTS

SUBROUTINE CUBIN(X,Y,N,A,B,C,D)

```

DIMENSION X(1),Y(1),A(1),B(1),C(1),D(1)
NM3 = N-3
DO 10 K=1,NM3
  K1 = K
  K2 = K+1
  K3 = K+2
  K4 = K+4
  AA = (Y(K1)*X(K2) - Y(K1)*X(K3) + Y(K2)*X(K3) - Y(K2)*X(K1) +
&   Y(K3)*X(K1)-Y(K3)*X(K2))/(X(K1)-X(K2))/(X(K2)-X(K3))/
&   (X(K1)-X(K3))
c
  BB = (Y(K2)*X(K3) - Y(K2)*X(K4) + Y(K3)*X(K4) - Y(K3)*X(K2) +
&   Y(K4)*X(K2) - Y(K4)*X(K3))/(X(K2) - X(K3))/(X(K3) - X(K4))/
&   (X(K2) - X(K4))
c
  A(K) = (AA-BB)/(X(K1)-X(K4))
  B(K) = AA-A(K)*(X(K1) + X(K2) + X(K3))
c
  C(K) = (Y(K1) - Y(K2))/(X(K1) - X(K2)) - B(K)*(X(K1) + X(K2)) -
&   A(K)*(X(K1)**2 + X(K1)*X(K2) + X(K2)**2)
10 D(K) = Y(K1) - C(K)*X(K1) - B(K)*X(K1)**2 - A(K)*X(K1)**3
c
  RETURN
  END

```

A.1.3 ANSYS to Experimental Results Interpolation Program :

```

PROGRAM ans_devflow
IMPLICIT none
INTEGER dim,r73_12,r73_30,r73_60,r73_90
INTEGER r125_12,r125_30,r125_60,r125_90
INTEGER r191_12,r191_30,r191_60,r191_90
PARAMETER (dim=2000,r73_12=28,r73_30=26,r73_60=22,r73_90=24)
PARAMETER (r125_12=29,r125_30=23,r125_60=25,r125_90=23)
PARAMETER (r191_12=27,r191_30=26,r191_60=27,r191_90=25)
INTEGER i,node,ncount,icount,ans,istn
REAL x,y,dum1,dum2,dum3,dum4,x_prof,y_loc,x_vel
REAL y_pos,x_out,dum5,dum6,re73_12,yre73_12,re73_30,yre73_30
REAL re73_60,yre73_60,re73_90,yre73_90,vel_fac,y_fac
REAL re125_12,yre125_12,re125_30,yre125_30,re125_60
REAL yre125_60,re125_90,yre125_90,re191_12,yre191_12,re191_30
REAL yre191_30,re191_60,yre191_60,re191_90,yre191_90
DIMENSION x(dim),y(dim),node(dim),x_out(dim)
DIMENSION re73_12(r73_12),yre73_12(r73_12),re73_30(r73_30)
DIMENSION yre73_30(r73_30),re73_60(r73_60),yre73_60(r73_60)
DIMENSION re73_90(r73_90),yre73_90(r73_90)
DIMENSION re125_12(r125_12),yre125_12(r125_12),re125_30(r125_30)
DIMENSION yre125_30(r125_30),re125_60(r125_60),yre125_60(r125_60)
DIMENSION re125_90(r125_90),yre125_90(r125_90)
DIMENSION re191_12(r191_12),yre191_12(r191_12),re191_30(r191_30)
DIMENSION yre191_30(r191_30),re191_60(r191_60),yre191_60(r191_60)
DIMENSION re191_90(r191_90),yre191_90(r191_90),x_vel(dim)
DIMENSION y_pos(dim)
CHARACTER *32 dum
DATA re73_12/0.,-0.5,-1.,-1.2,-1.2,-1.,-1.,-0.8,-0.5,0.,1.,
&   2.4,7.2,12.,17.8,22.,23.6,23.6,22.,18.2,15.,
&   11.,6.5,4.2,2.,1.,0./
DATA yre73_12/0.,.75,1.5,3.,4.,4.5,5.5,6.5,7.,8.,8.5,9.5,11.,
&   12.6,14.5,16.8,19.,21.5,24.,26.,28.9,30.2,31.5,
&   33.5,34.3,35.,35.5,36./
DATA re73_30/0.,-0.8,-1.8,-1.9,-1.5,-1.5,-1.1,-1.,-6,1.9,
&   3.9,7.,10.,13.9,17.5,21.,22.9,22.,20.,16.3,
&   12.2,8.9,5.1,3.,1.,0./
DATA yre73_30/0.,.5,1.5,2.5,3.,4.,4.5,5.5,7.,8.,9.5,11.,12.8,
&   14.5,16.5,19.5,21.5,24.,26.,28.5,30.2,31.5,33.5,
&   34.3,35.,36./

```

```

DATA re73_60/0.,2.,7.,8.,1.,2.,4.5,7.,11.,15.2,18.7,20.5,
& 21.,20.,17.,11.5,8.,5.2,3.,2.,1.,0./
DATA yre73_60/0.,1.5,3.,4.,4.5,6.5,8.,9.5,12.,14.3,16.8,19.,
& 21.5,23.5,26.,28.5,30.2,31.5,33.5,34.3,35.,36./
DATA re73_90/0.,1.,9.,1.2,2.2,3.,4.,4.5,6.9,9.,11.,14.3,17.5,
& 19.1,19.,18.,15.5,11.8,8.,5.3,3.,1.,5.0./
DATA yre73_90/0.,75,1.5,2.5,3.,4.,4.5,5.5,7.3,9.,10.5,12.5,15.,
& 17.5,20.,22.5,24.5,27.,29.4,31.,32.5,34.3,35.,36./
c
DATA re125_12/0.,-0.5,-1.,-1.,-1.5,-1.5,-1.2,-1.0,-0.75,
& 0.,0.5,1.,2.,3.3,7.2,11.8,16.,20.9,23.,24.,23.8,
& 20.5,17.,12.5,7.5,5.,2.,0./
DATA yre125_12/0.,0.8,1.5,2.5,3.2,4.,4.7,5.5,6.,7.,8.,8.5,9.5,
& 10.3,11.,12.5,14.,15.8,18.3,20.5,23.,25.5,27.5,
& 30.,31.5,33.2,34.2,35.,36./
DATA re125_30/0.,-0.5,-1.0,-2.,-1.8,-0.8,1.,3.,6.8,10.5,14.5,
& 18.2,22.,22.8,22.2,20.5,17.5,12.8,9.,5.,3.,1.5,0./
DATA yre125_30/0.,5,1.5,3.,5.,6.7,8.,9.5,11.,12.5,14.2,16.,
& 18.5,20.5,23.,25.2,27.6,30.3,31.8,33.5,34.2,
& 35.,36./
DATA re125_60/0.,-1.,-1.5,-1.8,-1.2,-1.,-0.5,0.8,2.5,6.,9.,13.,
& 16.,19.7,22.,21.2,20.3,17.5,14.,9.4,6.3,4.2,2.,
& 1.,0./
DATA yre125_60/0.,1.,1.5,2.5,3.2,4.,5.,5.7,7.,8.6,10.25,12.,
& 13.5,16.,18.5,20.7,23.,25.3,27.8,30.5,32.,33.5,
& 34.,35.,36./
DATA re125_90/0.,-0.5,-1.,-0.5,0.,0.8,1.2,4.5,9.,14.,17.5,20.,
& 20.2,19.2,17.,14.,12.,8.6,6.,3.,1.8,1.,0./
DATA yre125_90/0.,0.5,1.,1.8,2.5,3.3,4.5,6.5,9.,11.,13.5,16.,
& 18.3,20.5,23.,25.3,27.,28.5,30.3,32.,33.5,
& 34.3,36./
c
DATA re191_12/0.,-0.5,-0.8,-1.,-1.2,-1.2,-1.,-0.8,0.,1.,1.5,2.4,
& 6.5,11.5,16.5,21.,22.2,22.8,22.,20.2,16.6,12.5,
& 7.5,5.,2.,1.,0./
DATA yre191_12/0.,8,1.7,2.5,3.5,4.9,6.5,8.,8.7,9.5,10.5,11.2,
& 12.8,14.3,15.8,18.,20.5,23.,25.5,27.8,30.,31.5,
& 33.2,34.,34.5,35.5,36./
DATA re191_30/0.,-0.5,-1.,-1.6,-1.7,-1.6,-1.,-0.8,0.,1.3,2.4,
& 5.,9.2,13.2,17.3,21.5,22.8,22.9,22.,19.6,15.5,
& 11.2,6.5,4.,1.75,0./
DATA yre191_30/0.,8,1.5,2.3,3.5,5.,6.5,7.5,8.,8.6,9.6,11.3,12.7,
& 14.3,15.7,18.,20.5,23.,25.5,27.8,30.3,31.5,33.2,
& 34.,35.,36./
DATA re191_60/0.,-0.5,-1.2,-1.5,-2.,-1.5,-1.5,-1.4,-1.,0.,1.,
& 3.75,6.75,10.,14.2,19.,21.,22.,22.,20.,17.2,12.3,
& 8.2,4.4,2.75,1.2,0./
DATA yre191_60/0.,0.5,0.8,1.5,2.5,3.5,4.,5.,5.5,6.5,7.5,8.8,
& 10.5,11.8,13.5,16.,18.,20.5,23.,25.5,27.8,30.3,
& 32.,33.2,34.,35.,36./
DATA re191_90/0.,-0.25,-1.,-1.2,-1.2,-0.8,-0.3,0.5,2.5,5.5,10.5,
& 15.5,19.2,20.8,21.2,20.9,18.9,15.5,11.,7.2,4.,2.6,
& 1.5,0.8,0./
DATA yre191_90/0.,5,0.8,1.5,2.5,3.3,4.2,5.0,6.5,8.2,10.5,12.9,
& 15.,17.3,20.,22.,24.5,27.,29.5,31.,32.5,33.5,34.,
& 35.,36./
open(unit=11, file='loctn', status='unknown')
c
vel_fac = 16.0
y_fac = 36.0
c
----- Initialise Arrays -----
do i=1,dim
  node(i) = 0
  x_vel(i) = 0
  y_pos(i) = 0
enddo
write(6,9000)
read(5,9050) ans

```

```

write(6,9100)
read(5,*) istn
ncount = 1
10 continue
read(11,9150,end=20) dum
if (dum(4:7) .eq. 'NODE') then
  do i=1,20000
    read(11,*,end=20) node(ncount),x(ncount),y(ncount),
    &      dum1,dum2,dum3,dum4
    write(6,*) node(ncount),x(ncount),y(ncount)
    ncount = ncount + 1
  enddo
else
  goto 10
endif
20 continue
close(11)
icount = ncount - 1
c
open(unit=12, file='results.out',status='unknown')
if (ans .eq. 1) then
  write(12,9200)
  if (istn .eq. 1) then
    write(12,9250)
    do i=1,r73_12
      x_vel(i) = 4.85*re73_12(i)/vel_fac*0.001
      y_pos(i) = 45.0*yre73_12(i)/y_fac*0.001
    enddo
    call curfit(y_pos,x_vel,r73_12,y,x_out,icount,dum5,dum6,
    &      0,0,4,2,0,1,6)
  elseif (istn .eq. 2) then
    write(12,9300)
    do i=1,r73_30
      x_vel(i) = 4.85*re73_30(i)/vel_fac*0.001
      y_pos(i) = 45.0*yre73_30(i)/y_fac*0.001
    enddo
    call curfit(y_pos,x_vel,r73_30,y,x_out,icount,dum5,dum6,
    &      0,0,4,2,0,1,6)
  elseif (istn .eq. 3) then
    write(12,9350)
    do i=1,r73_60
      x_vel(i) = 4.85*re73_60(i)/vel_fac*0.001
      y_pos(i) = 45.0*yre73_60(i)/y_fac*0.001
    enddo
    call curfit(y_pos,x_vel,r73_60,y,x_out,icount,dum5,dum6,
    &      0,0,4,2,0,1,6)
  elseif (istn .eq. 4) then
    write(12,9400)
    do i=1,r73_90
      x_vel(i) = 4.85*re73_90(i)/vel_fac*0.001
      y_pos(i) = 45.0*yre73_90(i)/y_fac*0.001
    enddo
    call curfit(y_pos,x_vel,r73_90,y,x_out,icount,dum5,dum6,
    &      0,0,4,2,0,1,6)
  endif
elseif (ans .eq. 2) then
  write(12,9450)
c  read and scale for Reynolds Number equal 125
  if (istn .eq. 1) then
    write(12,9250)
    do i=1,r125_12
      x_vel(i) = 8.25*re125_12(i)/vel_fac*0.001
      y_pos(i) = 45.0*yre125_12(i)/y_fac*0.001
    enddo
    call curfit(y_pos,x_vel,r125_12,y,x_out,icount,dum5,dum6,
    &      0,0,4,2,0,1,6)
  elseif (istn .eq. 2) then
    write(12,9300)

```



```

do i=1,r125_30
  x_vel(i) = 8.25*re125_30(i)/vel_fac*0.001
  y_pos(i) = 45.0*yre125_30(i)/y_fac*0.001
enddo
call curfit(y_pos,x_vel,r125_30,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
elseif (istn .eq. 3) then
write(12,9350)
do i=1,r125_60
  x_vel(i) = 8.25*re125_60(i)/vel_fac*0.001
  y_pos(i) = 45.0*yre125_60(i)/y_fac*0.001
enddo
call curfit(y_pos,x_vel,r125_60,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
elseif (istn .eq. 4) then
write(12,9400)
do i=1,r125_90
  x_vel(i) = 8.25*re125_90(i)/vel_fac*0.001
  y_pos(i) = 45.0*yre125_90(i)/y_fac*0.001
enddo
call curfit(y_pos,x_vel,r125_90,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
endif
elseif (ans .eq. 3) then
write(12,9500)
c  read and scale for Reynolds Number equal 191
if (istn .eq. 1) then
write(12,9250)
do i=1,r191_12
  x_vel(i) = 12.7*re191_12(i)/vel_fac*0.001
  y_pos(i) = 45.0*yre191_12(i)/y_fac*0.001
enddo
call curfit(y_pos,x_vel,r191_12,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
elseif (istn .eq. 2) then
write(12,9300)
do i=1,r191_30
  x_vel(i) = 12.7*re191_30(i)/vel_fac*0.001
  y_pos(i) = 45.0*yre191_30(i)/y_fac*0.001
enddo
call curfit(y_pos,x_vel,r191_30,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
elseif (istn .eq. 3) then
write(12,9350)
do i=1,r191_60
  x_vel(i) = 12.7*re191_60(i)/vel_fac*0.001
  y_pos(i) = 45.0*yre191_60(i)/y_fac*0.001
enddo
call curfit(y_pos,x_vel,r191_60,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
elseif (istn .eq. 4) then
write(12,9400)
do i=1,r191_90
  x_vel(i) = 12.7*re191_90(i)/vel_fac*0.001
  y_pos(i) = 45.0*yre191_90(i)/y_fac*0.001
enddo
call curfit(y_pos,x_vel,r191_90,y,x_out,icount,dum5,dum6,
&      0,0,4,2,0,1,6)
endif
endif
do i=1,ncount-1
c  write(6,*) node(i),y(i),x_vel(i),x_out(i)
  write(12,9550) node(i),y(i),x_out(i)
enddo
close(12)
c
9000 format(//,'INPUT OPTION FOR REQUIRED REYNOLDS NUMBER',/,
&      '1. Re of 73',/, '2. Re of 125',/, '3. Re of 191')

```

```

9050 format(i1)
9100 format('INPUT OPTION FOR STATION NUMBER',/, '1. AT X = 12mm',/,
& '2. AT X = 30 mm',/, '3. AT X = 60 mm',/, '4. AT X = 90 mm')
9150 format(a32)
9200 format('RESULTS FOR REYNOLDS NUMBER EQUAL TO 73')
9250 format('LOCATION AT X = 12mm')
9300 format('LOCATION AT X = 30mm')
9350 format('LOCATION AT X = 60mm')
9400 format('LOCATION AT X = 90mm')
9450 format('RESULTS FOR REYNOLDS NUMBER EQUAL TO 125')
9500 format('RESULTS FOR REYNOLDS NUMBER EQUAL TO 191')
9550 format(i7,e16.7,2x,e16.7)
c
END

```

A.1.4 AWK Script Listing :

```

BEGIN { OFS="\t" }
$1 ~ /[0-9]+/ && NF==3 {
    print $3,$2;
    array[count]=$2;
    count++;
}
$1 ~ /[0-9]+/ && NF==2 {
    print $2, array[count2];
    count2++;
}
/^RES/
/^LOC/ {
    print;
    printf("\n\n");
    print "Exprmntl", "Rad Loc";
}
/NODE/&&/VX/ {
    printf("\n\n");
    print "Predicted", "Rad Loc";
}

```

A.2.0 Standard ANSYS Flotran Input Deck :

```

/BATCH
/NOPTINT
!
/COM ANSYS input file created by the MSC ANSYS input file translator (
/COM MSC/PATRAN ANSYS Version 6.0 ) on September 09, 1997 at 16:08:46.
!
/PREP7
!
!
/TITLE, Medium refined mesh of Backward Facing step (Mesh_2)
!
N, 1, 0.9, 0., 0.
N, 2, 0.9, 0.0025, 0.
N, 3, 0.9, 0.005, 0.
N, 4, 0.899999, 0.0075, 0.
N, 5, 0.9, 0.01, 0.
N, 6, 0.9, 0.0125, 0.
N, 7, 0.9, 0.015, 0.
N, 8, 0.9, 0.018, 0.
N, 9, 0.9, 0.021, 0.
N, 10, 0.899999, 0.024, 0.
...
...
...
...

```

```

N, 2871, -0.02, 0.027, 0.
!
/COM Defining Element Types
!
ET, 1, 141, 0, 0, 0, 0, 0
!
/COM Element props and connectivity for Region:Fluid_property
!
MAT, 1
TYPE, 1
EN, 1, 2843, 2842, 2855, 2856
EN, 2, 2842, 2841, 2854, 2855
EN, 3, 2856, 2855, 2864, 2865
EN, 4, 2855, 2854, 2861, 2864
EN, 5, 2841, 2840, 2850, 2854
EN, 6, 2840, 2835, 2834, 2850
EN, 7, 2854, 2850, 2849, 2861
EN, 8, 2850, 2834, 2833, 2849
EN, 9, 2835, 2821, 2820, 2834
EN, 10, 2821, 2805, 2804, 2820
...
...
...
EN, 2684, 2025, 2042, 2041, 2024
FINISH
/SOLUTION
!
FLDA, PROP, FLAG, 0
FLDA, ITER, EXEC, 3000
FLDA, OUTP, SUMF, 10
FLDA, PROP, DENS, 0
FLDA, PROP, VISC, 0
FLDA, SOLU, FLOW, T
FLDA, SOLU, TURB, F
FLDA, SOLU, COMP, F
FLDA, SOLU, TEMP, F
FLDA, OUTP, RESI, T
FLDA, PRIN, VX, F
FLDA, PRIN, VY, F
FLDA, PRIN, VZ, F
FLDA, PRIN, PRES, F
FLDA, PRIN, TEMP, F
FLDA, PRIN, ENKE, F
FLDA, PRIN, ENDS, F
FLDA, SOLU, SWRL, F
FLDA, QUAD, MOMD, 0
FLDA, QUAD, MOMS, 0
FLDA, QUAD, PRSD, 1
FLDA, QUAD, PRSS, 1
FLDA, QUAD, THRD, 0
FLDA, QUAD, THRS, 0
FLDA, QUAD, TRBD, 0
FLDA, QUAD, TRBS, 2
FLDA, PROT, DENS, CONSTANT
FLDA, PROP, UFRQ, 1
FLDA, VARY, DENS, F
FLDA, VARY, VISC, F
FLDA, VARY, COND, F
FLDA, PRIN, DENS, F
FLDA, PRIN, VISC, F
FLDA, PRIN, COND, F
FLDA, PRIN, EVIS, F
FLDA, PRIN, ECON, F
FLDA, OUTP, DEBG, 1
FLDA, TDMA, VX, 100
FLDA, TDMA, PRES, 100
FLDA, TDMA, TEMP, 100
FLDA, TDMA, ENKE, 10

```

FLDA, TDMA, ENDS,	10
FLDA, METH, PRES,	3
FLDA, MAXI, PRES,	1500
FLDA, METH, TEMP,	1
FLDA, MAXI, TEMP,	500
FLDA, SOLU, TRAN, F	
FLDA, TIME, NUMB,	10
FLDA, STEP, APPE,	0
FLDA, REST, OVER,	0
FLDA, OUTP, ERRR,	0
FLDA, SPEC, CONV, F	
FLDA, SPEC, ALGE,	2
FLDA, PROP, COND,	0
FLDA, PROP, SPHT,	0
FLDA, CAPP, VELO, F	
FLDA, CAPP, PRES, F	
FLDA, CAPP, TEMP, F	
FLDA, SOLU, SPEC, F	
FLDA, ITER, OVER,	0
FLDA, ITER, APPE,	0
FLDA, VARY, SPEC, F	
FLDA, TIME, GLOB,	20
FLDA, STEP, SUMF,	10
FLDA, TDMA, VY ,	100
FLDA, SRCH, PRES,	2
FLDA, TDMA, VZ ,	1
FLDA, SRCH, TEMP,	2
FLDA, METH, ENKE,	1
FLDA, METH, ENDS,	1
FLDA, MAXI, ENKE,	100
FLDA, SRCH, ENKE,	2
FLDA, METH, VX ,	1
FLDA, METH, VY ,	1
FLDA, METH, VZ ,	1
FLDA, MAXI, VX ,	100
FLDA, MAXI, VY ,	100
FLDA, SRCH, VX ,	2
FLDA, CONV, OUTP, BNOW	
FLDA, CONV, ITER,	1
FLDA, PROT, SPHT, CONSTANT	
FLDA, TURB, WALL, VAND	
FLDA, MAXI, VZ ,	100
FLDA, MAXI, ENDS,	100
FLDA, SRCH, ENDS,	2
FLDA, SRCH, VY ,	2
FLDA, SRCH, VZ ,	2
FLDA, PROT, VISC, CONSTANT	
FLDA, PROT, COND, CONSTANT	
FLDA, STEP, OVER,	0
FLDA, OUTP, VX , T	
FLDA, OUTP, VY , T	
FLDA, OUTP, VZ , F	
FLDA, OUTP, PRES, T	
FLDA, OUTP, ENKE, F	
FLDA, OUTP, ENDS, F	
FLDA, OUTP, TEMP, F	
FLDA, OUTP, SP01, F	
FLDA, OUTP, SP02, F	
FLDA, OUTP, SP03, F	
FLDA, OUTP, SP04, F	
FLDA, OUTP, SP05, F	
FLDA, OUTP, SP06, F	
FLDA, OUTP, DENS, F	
FLDA, OUTP, VISC, F	
FLDA, OUTP, COND, F	
FLDA, OUTP, EVIS, F	
FLDA, OUTP, ECON, F	
FLDA, OUTP, TTOT, F	

```

FLDA,  OUTP,  LMD1,  F
FLDA,  OUTP,  LMD2,  F
FLDA,  OUTP,  LMD3,  F
FLDA,  OUTP,  LMD4,  F
FLDA,  OUTP,  LMD5,  F
FLDA,  OUTP,  LMD6,  F
FLDA,  OUTP,  EMD1,  F
FLDA,  OUTP,  EMD2,  F
FLDA,  OUTP,  EMD3,  F
FLDA,  OUTP,  EMD4,  F
FLDA,  OUTP,  EMD5,  F
FLDA,  OUTP,  EMD6,  F
FLDA,  OUTP,  HFLU,  F
FLDA,  OUTP,  HFLM,  F
FLDA,  OUTP,  STRM,  T
FLDA,  OUTP,  MACH,  T
FLDA,  OUTP,  PTOT,  T
FLDA,  OUTP,  PCOE,  T
FLDA,  OUTP,  YPLU,  F
FLDA,  OUTP,  TAUW,  F
FLDA,  REST,  ITER,    0
FLDA,  REST,  NSET,    0
FLDA,  REST,  LSTP,    0
FLDA,  REST,  SSTP,    0
FLDA,  REST,  WFIL,  F
FLDA,  REST,  RFIL,  F
FLDA,  TERM,  PRES,  0.32000E-04
FLDA,  TERM,  TEMP,  0.10000E-07
FLDA,  NOMI,  SPHT,  -1.0000
FLDA,  GAMM,  COMP,  1.4000
FLDA,  NOMI,  DENS,  1000.0
FLDA,  COF1,  DENS,  0.00000E+00
FLDA,  COF2,  DENS,  0.00000E+00
FLDA,  COF3,  DENS,  0.00000E+00
FLDA,  NOMI,  VISC,  0.10000E-02
FLDA,  COF1,  VISC,  0.00000E+00
FLDA,  COF2,  VISC,  0.00000E+00
FLDA,  COF3,  VISC,  0.00000E+00
FLDA,  NOMI,  COND,  -1.0000
FLDA,  COF1,  COND,  0.00000E+00
FLDA,  COF2,  COND,  0.00000E+00
FLDA,  COF3,  COND,  0.00000E+00
FLDA,  TEMP,  NOMI,  293.00
FLDA,  TEMP,  BULK,  293.00
FLDA,  TEMP,  TTOT,  293.00
FLDA,  PRES,  REFE,  0.10135E+06
FLDA,  TURB,  SCTM,  1.0000
FLDA,  TURB,  SCTK,  1.0000
FLDA,  TURB,  SCTD,  1.3000
FLDA,  TURB,  SCTT,  1.0000
FLDA,  TURB,  ININ,  0.10000E-01
FLDA,  TURB,  INSF,  0.10000E-01
FLDA,  TURB,  RATI,  1000.0
FLDA,  TURB,  KAPP,  0.40000
FLDA,  TURB,  EWLL,  9.0000
FLDA,  TURB,  VAND,  26.000
FLDA,  TURB,  CMU,  0.90000E-01
FLDA,  TURB,  C1,  1.4400
FLDA,  TURB,  C2,  1.9200
FLDA,  RELX,  VX,  0.50000
FLDA,  RELX,  VY,  0.50000
FLDA,  RELX,  VZ,  0.50000
FLDA,  RELX,  PRES,  0.50000
FLDA,  RELX,  TEMP,  0.80000
FLDA,  RELX,  DENS,  0.50000
FLDA,  RELX,  VISC,  0.50000
FLDA,  RELX,  COND,  0.50000
FLDA,  STAB,  MOME,  0.10000E+16

```

```
FLDA, STAB, PRES, 0.10000E+16
FLDA, STAB, TURB, 0.10000E+16
FLDA, RELX, ENKE, 0.50000
FLDA, RELX, ENDS, 0.50000
FLDA, RELX, EVIS, 0.50000
FLDA, RELX, ECON, 0.50000
FLDA, CONV, PRES, 0.10000E-09
FLDA, CONV, TEMP, 0.10000E-06
FLDA, TIME, STEP, -1.0000
FLDA, TIME, TEND, 0.10000E+07
FLDA, TIME, APPE, 0.10000E+07
FLDA, SOLU, MIND, 0.10000E-05
FLDA, TOLR, TIME, 0.00000E+00
FLDA, TOLR, MASS, 0.00000E+00
FLDA, TOLR, DIST, 0.00000E+00
FLDA, TOLR, FANZ, 0.00000E+00
FLDA, STAB, VISC, 0.30000
FLDA, CAPP, UMIN, -0.10000E+21
FLDA, CAPP, UMAX, 0.10000E+21
FLDA, CAPP, VMIN, -0.10000E+21
FLDA, CAPP, VMAX, 0.10000E+21
FLDA, CAPP, WMIN, -0.10000E+21
FLDA, CAPP, WMAX, 0.10000E+21
FLDA, CAPP, PMIN, -0.10000E+07
FLDA, CAPP, PMAX, 0.10000E+07
FLDA, CAPP, TMIN, -0.10000E+21
FLDA, CAPP, TMAX, 0.10000E+21
FLDA, TIME, SUMF, 0.10000E+07
FLDA, TIME, PRES, 0.10000E-05
FLDA, TIME, TEMP, 0.10000E-05
FLDA, DELT, PRES, 0.10000E-09
FLDA, DELT, TEMP, 0.10000E-09
FLDA, CONV, ENKE, 0.10000E-04
FLDA, CONV, VX, 0.10000E-04
FLDA, TURB, BUC3, 0.00000E+00
FLDA, TURB, BUC4, 0.00000E+00
FLDA, TURB, BETA, 0.00000E+00
FLDA, BULK, BETA, 0.10000E+16
FLDA, SPEC, UGAS, 8314.3
FLDA, CONV, VY, 0.10000E-04
FLDA, CONV, VZ, 0.10000E-04
FLDA, CONV, ENDS, 0.10000E-04
FLDA, DELT, ENKE, 0.10000E-09
FLDA, DELT, ENDS, 0.10000E-09
FLDA, DELT, VX, 0.10000E-09
FLDA, DELT, VY, 0.10000E-09
FLDA, DELT, VZ, 0.10000E-09
FLDA, STAB, TEMP, 0.10000E+21
FLDA, TIME, OVER, 0.00000E+00
FLDA, REST, TIME, 0.00000E+00
FLDA, TURB, TRAN, 11.500
FLDA, RAD, BOLT, 0.00000E+00
FLDA, PROP, IVIS, 0.00000E+00
FLDA, MODV, VX, 0.00000E+00
FLDA, MODV, VY, 0.00000E+00
FLDA, MODV, VZ, 0.00000E+00
FLDA, MODV, PRES, 0.00000E+00
FLDA, MODV, ENKE, 0.00000E+00
FLDA, MODV, ENDS, 0.00000E+00
FLDA, MODV, TEMP, 0.00000E+00
FLDA, MODV, SP01, 0.00000E+00
FLDA, MODV, SP02, 0.00000E+00
FLDA, MODV, SP03, 0.00000E+00
FLDA, MODV, SP04, 0.00000E+00
FLDA, MODV, SP05, 0.00000E+00
FLDA, MODV, SP06, 0.00000E+00
FLDA, MODV, DENS, 0.00000E+00
FLDA, MODV, VISC, 0.00000E+00
```

```

FLDA, MODV, COND, 0.00000E+00
FLDA, MODV, EVIS, 0.00000E+00
FLDA, MODV, ECON, 0.00000E+00
FLDA, MODV, TTOT, 0.00000E+00
FLDA, MODV, LMD1, 0.00000E+00
FLDA, MODV, LMD2, 0.00000E+00
FLDA, MODV, LMD3, 0.00000E+00
FLDA, MODV, LMD4, 0.00000E+00
FLDA, MODV, LMD5, 0.00000E+00
FLDA, MODV, LMD6, 0.00000E+00
FLDA, MODV, EMD1, 0.00000E+00
FLDA, MODV, EMD2, 0.00000E+00
FLDA, MODV, EMD3, 0.00000E+00
FLDA, MODV, EMD4, 0.00000E+00
FLDA, MODV, EMD5, 0.00000E+00
FLDA, MODV, EMD6, 0.00000E+00
FLDA, MODR, VX, 0
FLDA, MODR, VY, 0
FLDA, MODR, VZ, 0
FLDA, MODR, PRES, 0
FLDA, MODR, ENKE, 0
FLDA, MODR, ENDS, 0
FLDA, MODR, TEMP, 0
FLDA, MODR, SP01, 0
FLDA, MODR, SP02, 0
FLDA, MODR, SP03, 0
FLDA, MODR, SP04, 0
FLDA, MODR, SP05, 0
FLDA, MODR, SP06, 0
FLDA, MODR, DENS, 0
FLDA, MODR, VISC, 0
FLDA, MODR, COND, 0
FLDA, MODR, EVIS, 0
FLDA, MODR, ECON, 0
FLDA, MODR, TTOT, 0
FLDA, MODR, LMD1, 0
FLDA, MODR, LMD2, 0
FLDA, MODR, LMD3, 0
FLDA, MODR, LMD4, 0
FLDA, MODR, LMD5, 0
FLDA, MODR, LMD6, 0
FLDA, MODR, EMD1, 0
FLDA, MODR, EMD2, 0
FLDA, MODR, EMD3, 0
FLDA, MODR, EMD4, 0
FLDA, MODR, EMD5, 0
FLDA, MODR, EMD6, 0
BFUNIF,TEMP,_TINY
AUTOTS,OFF
NSUBST, 0, 0, 0,OFF
KBC, 0
KUSE, 0
TIME, 0.000000000E+00
TREF, 0.000000000E+00
ALPHAD, 0.000000000E+00
BETAD, 0.000000000E+00
DMPRAT, 0.000000000E+00
TIMINT,OFF,STRU
TIMINT,OFF,MAG
CNVTOL,F,-1
CNVTOL,U,-1
CNVTOL,M,-1
CNVTOL,ROT,-1
CNVTOL,FLUX,-1
CNVTOL,MAG,-1
CNVTOL,CSG,-1
CNVTOL,A,-1
CRPLIM, 0.100000000

```

```

NCNV, 1, 0.00000000E+00, 0, 0.00000000E+00, 0.00000000E+00
LNSRCH,OFF
NEQIT, 0
PRED,OFF,OFF
ERESX,DEFA
ACEL, 0.00000000E+00, 0.00000000E+00, 0.00000000E+00
OMEGA, 0.00000000E+00, 0.00000000E+00, 0.00000000E+00, 0
DOMEGA, 0.00000000E+00, 0.00000000E+00, 0.00000000E+00
CGLOC, 0.00000000E+00, 0.00000000E+00, 0.00000000E+00
CGOMEGA, 0.00000000E+00, 0.00000000E+00, 0.00000000E+00
DCGOMG, 0.00000000E+00, 0.00000000E+00, 0.00000000E+00
IRLF, 0
/COM Boundary Condition at Wall
!
NSEL,S,LOC,Y,0.045,0.045
D,ALL,VX,0.
D,ALL,VY,0.
NSEL,ALL
NSEL,S,LOC,Y,0.0,0.0
D,ALL,VX,0.
D,ALL,VY,0.
NSEL,S,EXT
NSEL,R,LOC,X,9,9
D,ALL,PRES,0.
NSEL,ALL
NSEL,S,EXT
NSEL,R,LOC,X,0,0
D,ALL,VX,0.
D,ALL,VY,0.
NSEL,ALL
NSEL,S,EXT
NSEL,R,LOC,Y,0.015,0.015
NSEL,R,LOC,X,-0.02,0.0
D,ALL,VX,0.
D,ALL,VY,0.
NALL
EALL
!
!
/COM Inlet Velocity Profile for Reynolds Number=73
!
/COM D, 2811,VX, 0.000000E+00
/COM D, 2826,VX, 0.2271414E-02
/COM D, 2839,VX, 0.4192654E-02
/COM D, 2843,VX, 0.000000E+00
/COM D, 2853,VX, 0.5904872E-02
/COM D, 2856,VX, 0.2789930E-02
/COM D, 2863,VX, 0.6912759E-02
/COM D, 2865,VX, 0.4932852E-02
/COM D, 2869,VX, 0.7274998E-02
/COM D, 2870,VX, 0.6216876E-02
/COM D, 2871,VX, 0.6996886E-02
!
!
/COM Inlet Velocity Profile for Reynolds Number=125
!
D, 2811,VX, 0.000000E+00
D, 2826,VX, 0.4199173E-02
D, 2839,VX, 0.7975169E-02
D, 2843,VX, 0.000000E+00
D, 2853,VX, 0.1031250E-01
D, 2856,VX, 0.5070239E-02
D, 2863,VX, 0.1131554E-01
D, 2865,VX, 0.8844454E-02
D, 2869,VX, 0.1184634E-01
D, 2870,VX, 0.1072569E-01
D, 2871,VX, 0.1160818E-01
!

```



```
!  
/COM Inlet Velocity Profile for Reynolds Number=191  
!  
/COM D, 2811,VX, 0.0000000E+00  
/COM D, 2826,VX, 0.6394044E-02  
/COM D, 2839,VX, 0.1233992E-01  
/COM D, 2843,VX, 0.0000000E+00  
/COM D, 2853,VX, 0.1539515E-01  
/COM D, 2856,VX, 0.8861763E-02  
/COM D, 2863,VX, 0.1710776E-01  
/COM D, 2865,VX, 0.1438829E-01  
/COM D, 2869,VX, 0.1778000E-01  
/COM D, 2870,VX, 0.1674677E-01  
/COM D, 2871,VX, 0.1784879E-01  
/GO  
FINISH  
/SOLU  
SOLVE
```