

**EFFICIENT INVERSE METHODS FOR SUPERSONIC AND
HYPERSONIC BODY DESIGN, WITH LOW WAVE DRAG ANALYSIS**

by

Jaewoo Lee

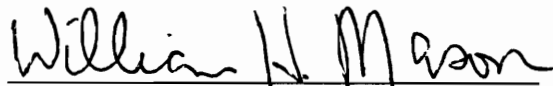
Dissertation submitted to the Faculty of the
Virginia Polytechnic Institute and State University
in partial fulfillment of the requirements for the degree of

DOCTOR OF PHILOSOPHY

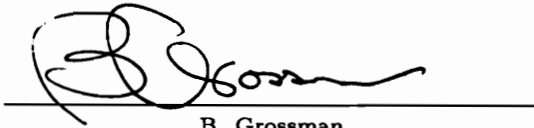
in

Aerospace Engineering

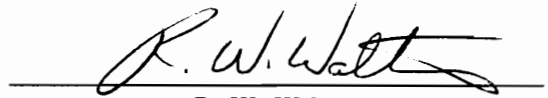
APPROVED:



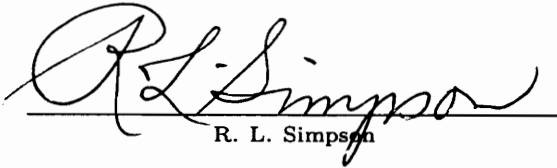
W. H. Mason, Chairman



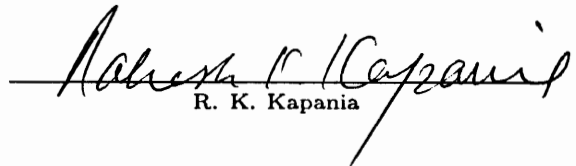
B. Grossman



R. W. Walters



R. L. Simpson



R. K. Kapania

April, 1991

Blacksburg, Virginia

EFFICIENT INVERSE METHODS FOR SUPERSONIC AND HYPERSONIC BODY DESIGN, WITH LOW WAVE DRAG ANALYSIS

by

Jaewoo Lee

Committee Chairman: William H. Mason

Aerospace Engineering

(ABSTRACT)

With the renewed interest in the supersonic and hypersonic flight vehicles, new inverse Euler methods are developed in these flow regimes where a space marching numerical technique is valid. In order to get a general understanding for the specification of target pressure distributions, a study of minimum drag body shapes was conducted over a Mach number range from 3 to 12. Numerical results show that the power law bodies result in low drag shapes, where the $n=.69$ ($l/d = 3$) or $n=.70$ ($l/d = 5$) shapes have lower drag than the previous theoretical results ($n=.75$ or $n=.66$ depending on the particular form of the theory). To validate the results, a numerical analysis was made including viscous effects and the effect of gas model. From a detailed numerical examination for the nose regions of the minimum drag bodies, aerodynamic bluntness and sharpness are newly defined.

Numerous surface pressure-body geometry rules are examined to obtain an inverse procedure which is robust, yet demonstrates fast convergence. Each rule is analyzed and examined numerically within the inverse calculation routine for supersonic ($M_\infty = 3$) and hypersonic ($M_\infty = 6.28$) speeds. Based on this analysis, an inverse method for fully three dimensional supersonic and hypersonic bodies is developed using the Euler equations. The method is designed to be easily incorporated into existing analysis codes, and provides the aerodynamic designer with a powerful tool for design of aerodynamic shapes of arbitrary cross section. These shapes can correspond to either "wing like" pressure distributions or to "body like" pressure distributions. Examples are presented illustrating the method for a

non-axisymmetric fuselage type pressure distribution and a cambered wing type application. The method performs equally well for both nonlifting and lifting cases. For the three dimensional inverse procedure, the inverse solution existence and uniqueness problem are discussed. Sample calculations demonstrating this problem are also presented.

ACKNOWLEDGEMENT

The author would like to express his deepest appreciation for the invaluable advice, support and encouragement given to him by his advisor, Dr. William H. Mason. Without Dr. Mason's enthusiasm and proper guidance, it would have been difficult to accomplish this work.

The author would also like to thank Dr. Robert W. Walters for providing him with access to `cf13de` and consulting with him on the use of the code, and for his critique of this dissertation. The author would like to thank Dr. Bernard Grossman, Dr. Roger L. Simpson and Dr. Rakesh K. Kapania for serving on the Advisory Committee and their comments and suggestion on this study.

The author would also like to thank all of his friends and peers for their daily help and innumerable discussions during the study.

The author would like to thank his parents for all of the thoughtful considerations and support provided throughout his education. Finally, the author would like to thank his wife Hyewon, his daughter Sangeun, and his newborn son Sangho for their support and patience during his study.

TABLE OF CONTENTS

ABSTRACT	ii
ACKNOWLEDGEMENT	iv
LIST OF FIGURES AND TABLES	viii

I. INTRODUCTION

1. Supersonic/Hypersonic Minimum Drag Bodies	1
2. Aerodynamic Design; Inverse Methods and Numerical Optimization	3
2.1 Inverse Methods in Subsonic and Transonic Flow Regimes	5
2.2 Inverse Methods in Supersonic and Hypersonic Flow Regimes	7
2.3 The Existence and Uniqueness of the Inverse Solutions	8
3. The Objective of the Present Study	10

II. MINIMUM DRAG BODIES IN SUPERSONIC AND HYPERSONIC FLOW REGIMES

1. Review of Some Previous Minimum Drag Body Studies	12
2. Numerical Investigation	16
2.1 Computational Methodology and Grid Selection	17
2.2 Numerical Accuracy	18
3. Results for Optimum Supersonic/Hypersonic Bodies	19
3.1. Power Law Bodies, $l/d=3$	19
3.2. Power Law Bodies, $l/d=5$	21
3.3. Thermodynamic Gas Effects	21

- 3.4. Numerical Examination of the Nose Region 21
- 3.5. Effects of Viscosity 22
- 3.6. The Hayes Forebody Shape 22
- 3.7. Discussion of the Results of Optimum Body Shape 23

III. AERODYNAMICALLY SHARP AND BLUNT BODIES

- 1. The Issue: A Discovery during the Analysis of Minimum Drag Shapes 24
- 2. Geometrical Analysis for Several Minimum Drag Shapes 25
- 3. Newtonian Flow/cfl3de Analysis 28
- 4. Implications for Nose Shaping 30

IV. INVERSE DESIGN METHOD FOR AXISYMMETRIC BODIES

- 1. Description of The Method 31
 - 1.1. General Approach 31
 - 1.2. Surface Pressure and Body Geometry Relations 34
 - 1.3. Adaptation to Incorporate Root Finding Scheme 39
- 2. Numerical Evaluation for Axisymmetric Bodies 40
 - 2.1. Hypersonic Minimum Drag Body 41
 - 2.2. Test Cases Having Quadratically Varying Target Pressures 41
 - 2.3. Test Case IV 46
 - 2.4. Ogive Cylinder 47

V. INVERSE DESIGN METHOD FOR THREE-DIMENSIONAL BODIES

- 1. Description of The Method 50
 - 1.1. General Approach 50
 - 1.2. Method in Detail 52

1.3. Method of Accelerating the Convergence	56
2. Example Calculations and Results	57
2.1. Elliptic Cone Test Case ($a/b = 2.0$)	59
2.2. "Wing Like" Nonconical Cambered Body	60
 VI. INVERSE SOLUTION UNIQUENESS AND THE DOMAIN OF ITS EXISTENCE	
1. Domain of Inverse Solution Existence	63
2. Inverse Solution Uniqueness	65
 VII. CONCLUDING REMARKS.	71
 REFERENCES	75
FIGURES	83
TABLES	140
APPENDICES	
Appendix A: Details of Analysis Code Application to the Minimum Drag Body Study	149
Appendix B: Computational Grids Used in the Study	151
Appendix C: About the Inverse Code	154
VITA	159

LIST OF FIGURES AND TABLES

FIGURES

II. Minimum Drag Bodies in Supersonic and Hypersonic Flow Regimes

Figure 1. Demonstration of “Gap” between Linearized Supersonic and Hypersonic
Minimum Drag Body Theory

Figure 2. Details of Blunt Nose Shape

Figure 3. Comparison of Minimum Drag Shapes

Figure 4. Computational Grid Used in the Minimum Drag Body Study

Figure 5. Convergence of Drag with Increasing Grid Density

Figure 6. Convergence of Pressure Distribution with Increasing Grid Density

Figure 7. Numerical Determination of Optimum “ n ” for Power Law Shapes

Figure 8. Pressure Distributions for Several Power Law Shapes

Figure 9. Mach Number Effects for Several Shapes

Figure 10. Comparison of $l/d = 3.0$ & 5.0

Figure 11. Comparison of Pressure Distributions of Perfect Gas and Equilibrium Air

Figure 12. Comparison of Temperature Distributions between Perfect Gas and Equilibrium
Air

Figure 13. Comparison of Pressure Distributions for Different Numerical Schemes

Figure 14. Comparison of the Shock Locations for the Different Numerical Schemes

Figure 15. Pressure and Skin Friction Distribution for the Optimum Power Law Shape

Figure 16. Comparison of Power Law Shapes and Pressure Distributions with Hayes’
Body Shape

Figure 17. Results of Minimum Drag Body Study Added to the Fink⁶⁰ Chart

III. Aerodynamically Blunt and Sharp Bodies

Figure 18. Comparison of Power Law Body ($n=0.75$) with Inscribed Circle near the Nose

Figure 19. Curvature Distribution of Various Power Law Bodies near the Nose

Figure 20. Newtonian Solutions and Numerical Result for Power Law Bodies; C_p vs. s

Figure 21. Newtonian Solutions and Numerical Result for Power Law Bodies; C_p vs. x

Figure 22. Newtonian Solutions for Power Law Bodies; dC_p/ds vs. s

Figure 23. Newtonian Solutions for Power Law Bodies; dC_p/ds vs. *power n*

IV. Inverse Design Method for Axisymmetric Bodies

Figure 24. Flow Chart of Inverse Method for Axisymmetric Bodies

Figure 25. Comparison of $A(M_L, \theta)$ in Four Pressure-Shape Rules ($M_\infty = 3.0$)

Figure 26. Comparison of $A(M_L, \theta)$ in Four Pressure-Shape Rules ($M_\infty = 6.0$)

Figure 27. Axial Computational Grids for Axisymmetric Inverse Test Cases

Figure 28. Test Target Pressure Distributions

Figure 29. Test Target Pressure I and Resulting Body Shape

Figure 30. Test Target Pressure II and Resulting Body Shape

Figure 31. Test Target Pressure III and Resulting Body Shape

Figure 32. Test Target Pressure IV and Resulting Body Shape

Figure 33. Pressure and Body Geometry of Secant Ogive Cylinder ($l/d = 3.0$)

V. Inverse Design Method for Three Dimensional Bodies

Figure 34. Flow Chart of Three Dimensional Inverse Procedure

Figure 35. Three Dimensional Inverse Procedure in Detail

Figure 36. Body Geometry Correction along the Surface Streamline

Figure 37. Comparison with Experimental Result for the Elliptic Cone

- Figure 38. Convergence Behavior of the Elliptic Cone Test Case for Different Grid Densities and Regula-Falsi Option (Initial $Y_{max} = 0.26663$)
- Figure 39. Convergence Behavior of the Elliptic Cone Test Case for Different Grid Densities and Regula-Falsi Option (Initial $Y_{max} = 0.4$)
- Figure 40. Body Geometry Convergence of the Elliptic Cone Test Case ($M_{\infty}=6.28, \alpha=0^{\circ}$)
- Figure 41. Comparison of Convergence Behavior for Two Surface Pressure-Body Geometry Rules
- Figure 42. Body Geometry Convergence of the Elliptic Cone Test Case ($M_{\infty}=6.28, \alpha=10^{\circ}$)
- Figure 43. Pressure Coefficient Convergence of the Elliptic Cone Test Case ($M_{\infty} = 6.28, \alpha = 10^{\circ}$)
- Figure 44. Convergence Behavior of the Elliptic Cone Test Case with and without Regula-Falsi ($M_{\infty}=6.28, \alpha = 10^{\circ}$)
- Figure 45. Body Geometry Convergence of the Elliptic Cone Test Case ($M_{\infty}=3.0, \alpha=0^{\circ}$)
- Figure 46. Convergence Behavior of the Elliptic Cone Test Case with and without Regula-Falsi ($M_{\infty}=3.0, \alpha = 0^{\circ}$)
- Figure 47. Cross-Flow Plane Computational Grids for Cambered Body at $x/l = 1.0$
- Figure 48. Body Geometry Convergence of the Cambered Body ($M_{\infty} = 6.28, \alpha = 0^{\circ}$)
- Figure 49. Convergence Behavior of the 'Wing Like' Nonconical Cambered Body with and without Regula-Falsi Scheme
- Figure 50. Body Geometry Convergence of the Cambered Body ($M_{\infty} = 6.28, \alpha = 10^{\circ}$)
- Figure 51. Pressure Coefficient Convergence of the Cambered Body ($M_{\infty} = 6.28, \alpha=10^{\circ}$)
- Figure 52. Convergence Behavior of the 'Wing Like' Nonconical Cambered Body with and without Regula-Falsi

VI. Inverse Solution Uniqueness and the Domain of Its Existence

Figure 53. Domain of Existence (Elliptic Cone: $M_\infty=6.28$, $\alpha = 0^\circ$)

Figure 54. Domain of Existence (Elliptic Cone: $M_\infty=6.28$, $\alpha = 10^\circ$)

Figure 55. Domain of Existence (Cambered Body: $M_\infty=6.28$, $\alpha = 0^\circ$)

Figure 56. Pressure Coefficient as a Function of Surface Slope for Several Pressure-Shape Rules

Figure 57. Effect of Fixed Planform:

Body Geometry Convergence of Elliptic Cone ($M_\infty=6.28$, $\alpha = 0^\circ$)

Figure 58. ‘Wing Like’ Nonconical Cambered Body with Different Semispan Lengths

Figure 59. Several C_p Distributions with Different Semispan Lengths in ‘Wing Like’ Nonconical Cambered Body

Figure 60. Inverse Solution Uniqueness:

Resulting Body Geometries from Different Data Planes ($M_\infty=6.28$, $\alpha =10^\circ$)

TABLES

Table 1. Summary of the Various Minimum Drag Shapes

Table 2. Drag Coefficient of the Power Law Bodies with different Power n

Table 3. Comparison of Drag Coefficients between Perfect Gas and Equilibrium Air Model

Table 4. Convergence Behavior of Several Pressure- Shape Rules for Test Case I

Table 5. Convergence Behavior of Several Pressure- Shape Rules for Test Case II

Table 6. Convergence Behavior of Several Pressure- Shape Rules for Test Case III

Table 7. Drag Coefficients of the Axisymmetric Inverse Test Cases

I. INTRODUCTION

Aerodynamic *analysis* methods have progressed very rapidly with the development of computational fluid dynamics. With the interest in new design concepts for both the High Speed Civil Transport at supersonic speeds and X-30 National Aerospace Plane at hypersonic speeds as means of beginning a new era of advanced vehicles¹, practical aerodynamic *design* methods for this speed range are required. While both supersonic and hypersonic configurations have been extensively studied in the past, the possibility of such configurations becoming practical relies on achieving the potential of advanced propulsion and materials technology, as well as aerodynamic design. Based on the history of propulsion advances, it can be expected that the first generation of these new propulsion systems will not have reached full maturity, and that performance may be initially marginal. Thus, not only is the propulsion system crucial, but a maximum effort will be required to provide the minimum drag airframe².

1. Supersonic/Hypersonic Minimum Drag Bodies

Minimum drag bodies have been studied extensively both for supersonic and hypersonic flight, with an entire book³ devoted to the subject during a previous period of intense interest. At that time the flow fields were modeled with simplified theories and theoretical analysis was used to determine the minimum drag shapes. Subsequent to that time, interest in minimum drag shapes received less attention (with exceptions), while other design considerations dominated the aerodynamic design problem. However, with sustained cruise flight once again of interest, the importance of achieving low drag levels dictates that additional work be done in this area. Moreover, a study for minimum drag bodies provides insight into the most desirable choice for the surface pressure distributions.

The proper specification of surface pressure distribution plays a key role in the development of practical aerodynamic design methods, especially for inverse methods. As will be explained later, the selection of the actual target pressure distribution is directly related to the existence and uniqueness of the inverse solution and must be based on previous design experience. Generally this amounts to selecting a pressure distribution that controls adverse viscous effects and, especially in the supersonic and hypersonic case, minimizes drag. From this point of view, it is valuable to reexamine and investigate the minimum drag bodies in the supersonic and hypersonic flow regimes with the advanced computational design tools. Since the original minimum drag shapes were derived, a revolution has taken place in the area of aerodynamics. Body shapes can now be analyzed routinely using advanced computational methods. This affords us the opportunity to reexamine the classical minimum drag shapes, thus to provide a clarification of the historical results for hypersonic minimum drag shapes through a computational examination of several of the previously proposed minimum drag shapes using an advanced computational method. The interest is in the high supersonic/moderate hypersonic speed regime ($M_\infty = 3\sim 12$, where the high Mach number is of interest for low drag and hence slender shapes), where neither the linearized supersonic nor the Newtonian hypersonic analysis will be accurate. This is precisely the Mach number range of interest in a number of current aircraft studies.

A geometric property of many minimum drag shapes in this Mach number range, is the surprising feature that the shape of the axisymmetric body, $r(x)$ has initial slope dr/dx , of 90° . This means that a supersonic or hypersonic minimum drag shape is blunt. This is consistent with the fact that a slight bluntness near the nose is essential in hypersonic flow to decrease not only pressure drag but also friction drag and the amounts of heat transfer to the body surface. In some cases, a geometrically blunt body is not necessarily an aerodynamically blunt body⁴. Thus, during a numerical investigation for the supersonic and hypersonic minimum drag bodies, specific attention will be paid to the examination of the nose region to investigate the effect of the bluntness of the nose on the flowfield solutions.

2. Aerodynamic Design; Inverse Methods and Numerical Optimization

Even though computational fluid dynamics is regarded as the primary aerodynamic design tool for some design problems⁵, aerodynamic design methods have not received the same level of attention that the analysis methods have received. Candidate aerodynamic design methods are generally divided into two categories; inverse methods and numerical optimization using direct analysis methods repetitively. The two approaches each have strengths and weaknesses, and are perhaps best used together for current design projects. In particular, inverse methods can be used to generate the design variables or 'shape functions' required for numerical optimization. This approach is part of the 'Smart Aerodynamic Optimization' concept described in reference 6.

Numerical optimization methods are characterized by the use of automated design procedures in which an optimization algorithm and a fluid dynamics solver are linked together to, directly, minimize a given aerodynamic objective function, such as drag. The important parts of the numerical optimization approach are the choices of an aerodynamic objective function and constraints, a set of design variables and shape functions, and proper numerical optimization algorithm⁷. The choice of the shape functions which is related to the determination of the geometry to be designed and corresponding design variables, is of utmost importance because the computational cost is directly dependent upon the number of design variables. Hicks *et al.* used simple polynomial expressions (4th degree polynomial with a square root leading term) in early applications^{8,9}. In later applications^{10,11,12}, they used a more sophisticated class of shape functions (a set of geometric shape functions), which enable multiple design point problems to be solvable. Vanderplaats¹² selected four existing airfoils as shape functions and obtained the desired airfoil geometry by the linear superposition of these four airfoils. A reasonable and refined choice for the shape functions is suggested by Aidala *et al.*⁶. They proposed shape functions based on pressure distribution changes using an inverse program. These shape functions are aerodynamically more meaningful, in comparison to the physically

meaningless bumps or polynomials used in the previous optimization studies. While the choice of the design variables is of great practical significance, the choice of the objective function, in conjunction with the choice of the aerodynamic and geometric constraints, is also of fundamental and practical interest. Typical selection of the objective function has been wave drag^{8,9,10,14}.

Even though most efforts have been given to the development of the numerical optimization method in transonic potential flow, there are a few optimization techniques developed for the Euler equation in the supersonic and hypersonic speeds. Using the widely used optimization code CONMIN¹⁵ developed by Vanderplaats, Bock¹⁶ designed the minimum drag body using the Euler space marching program for bodies of revolution at supersonic speed ($M_\infty=3.0$). Schone¹⁷ applied the numerical optimization technique to the design of wings (conical or three dimensional wings) in the supersonic speed ($M_\infty=4.8$) using an analysis code for the Euler equation. Using modified Newton theory, Dulikravich *et al.*¹⁸ designed a minimum drag body at zero incidence in hypersonic flow. Anderson and co-workers^{19,20} designed viscous optimized hypersonic waveriders using the non-linear simplex method.

Because of the excessively large computational costs, at least, in three dimensional flow, the possibility of employing numerical optimization methods arose only after significant computer power became generally available. Thus these methods are the most recent, most expensive and possibly the most promising ones in the long run. Perhaps the key advantage is the prospect of finding an optimal body shape directly. However, these methods depend critically on the user assumed form of the answer (both in selecting the design variables and the objective function), and, moreover, local rather than global optima can appear during the optimization.

The other approach to aerodynamic design is the inverse method, where the pressure distribution is specified and the analysis determines the required geometry. Inverse methods are generally much cheaper to use than optimization methods especially for the

design of three-dimensional body shapes, although optimum designs do not arise directly.

Even experienced designers may not always know the optimum, or most desirable pressure distribution on a surface, but they can usually recognize an undesirable distribution that must be modified²¹. Thus, the tedious cut and try shaping has been replaced by the inverse technique: Rather than guessing shapes that must be assessed by analysis or testing, inverse methods allow our understanding of flow physics to directly impact the course of design⁵: The inverse method has been considered important for decades because many desirable features of the flow field, such as minimization of the wave drag, and delay of boundary layer separation and laminar-to-turbulent transition can be achieved by proper prescription of the pressure distribution along the surface of the as yet unknown body^{22,23}. Over the past few years at Boeing, the inverse method has been the primary design tool for transonic wing development^{5,21}.

2.1. Inverse Methods in Subsonic and Transonic Flow Regimes

The aerodynamic design methods in these flow regimes have been reviewed thoroughly by Slooff⁷. His classification for the subsonic and transonic inverse methods has been adopted as follows.

Numerous inverse design procedures have been developed, but most of the effort has been directed toward the airfoil and wing design at subsonic free stream speeds assuming the potential flow. Currently existing inverse design methods for these speed ranges can be classified into two main categories: Methods utilizing *Dirichlet-type* boundary conditions derived from the target pressure distribution; Methods utilizing *Neumann-type* boundary conditions in combination with some geometry correction procedure (residual-correction method).

In the *Dirichlet-type* approach, airfoil geometry corrections are derived by

integrating - either explicitly, or in some implicit manner - the transpiration mass flow over airfoil shape estimates²⁴. The main advantage of the *Dirichlet-type* method is rapid convergence, provided appropriate auxiliary algebraic constraint relations for airfoil closure and regularity are satisfied (there will be a detailed explanation of these constraints in a later part of this chapter). Zedan and Dalton²⁵ developed an inverse method for axisymmetric bodies in incompressible potential flow. Johnson and Rubbert²⁶ applied this approach at subsonic flow using PANAIR panel program. Reference 26 does not provide details about the procedure utilized for updating the geometry from the normal velocity components in each iteration. The first reported effort to solve the full potential transonic inverse airfoil problem using the *Dirichlet-type* approach is that of Tranen²⁷. Volpe²⁸ and Arlinger²⁹ used approaches similar to Tranen's in designing two element airfoil systems. A distinctly different approach has been taken by Carlson³⁰. Instead of using a body conforming finite difference mesh in the circle plane, Carlson uses Cartesian coordinates in the physical plane. The first inverse method for three dimensional transonic potential flows is that of Henne³¹. His method can be considered as the three dimensional equivalent of Tranen's²⁷ two dimensional method. Because Henne does not address the complete three dimensional problem of determining the velocity components, it is doubtful whether the scheme will converge in the proper mathematical sense⁷.

In the *residual-correction type* approach, a sequence of analysis problem is solved over a corresponding series of geometries. Each geometry is provided by some rational method depending on the difference between the computed pressure and prescribed pressure being driven to zero. The main advantage of the *residual-correction type* approach is its simplicity. Only a small investment is required, because any existing analysis code can be used without modification. However, it is known that this approach has a modest rate of convergence. A simple, but apparently effective *residual-correction type* approach has been described by Davis³², who suggested a simple body surface alteration procedure. Using simple wavy-wall formula, he developed a design method for the supersonic wing and transonic airfoil. Another possibility using this approach has been studied by Garabedian and McFadden³³ for three dimensional wings at transonic speeds.

Malone³⁴ used PANAIR panel code to design three dimensional wing at subsonic flow. Malone *et al.*³⁵ modified the Garabedian and McFadden's³³ original inverse design procedure and designed two dimensional airfoils and three dimensional nacelle configurations at transonic speeds. Takanashi³⁶ and Tatsumi and Takanashi³⁷ used the residual-correction concept combined with existing three dimensional transonic full potential flow analysis code to design swept wings. A method for axisymmetric, semi-infinite inlets has been described by Ives³⁸.

All the inverse methods described above have been developed for the airfoils or wings in the subsonic and transonic potential flows, which are essentially valid even for the rotational flow with relatively weak shock ($M_n < 1.2 \sim 1.3$)³⁹. But as pointed out by Moretti⁴⁰, it seems to be necessary to abandon the potential equation and to replace it with full Euler equation in the presence of a shock which is not weak, because the potential flow calculation may provide incorrect flow field solutions⁴¹. Inverse methods for the Euler equations have been developed for airfoils and cascades by Giles *et al.*^{42,43}, for axisymmetric duct and blade cascade by Meauze⁴⁴ and three dimensional extension of reference 44 by Couaillier and Veuillot⁴⁵ in transonic flow. Lin *et al.*²¹ designed three dimensional transonic nacelles and winglets using the *residual-correction type* approach.

2.2. Inverse Methods in Supersonic and Hypersonic Flow Regimes

Much less effort has been devoted to the development of inverse methods for supersonic and hypersonic bodies. This is especially true for three dimensional bodies. In particular, the Douglas Aircraft study⁴⁶ of supersonic cruise airplanes identified the need for an inverse method for HSCT class vehicles, which will allow the aerodynamic designer to specify a pressure distribution consistent with the requirements for maintaining laminar flow.

Important considerations in supersonic/hypersonic design are the use of methods that

exploit the space marching techniques available for high speed flow, and the issue of determining the planform shape as well as body contours within a specified planform outline. The most recent work appears to be the methods developed by Sirovich and co-workers^{47,48}. They developed the inverse method for two dimensional airfoil and axisymmetric bodies in the supersonic flow using the method of characteristics. By developing the body slope correction procedure using Lighthill's approximate solution of the linear equation for the axial velocity, Barger⁴⁹ designed a supersonic body of revolution from a prescribed pressure distribution. However, it appears that the approach developed by Davis³² provides the most fruitful avenue of development as described above. The reason for adopting this approach is the large amount of effort expended on analysis methods which can be used directly, together with the general tendency of direct analysis methods to be more robust than inverse methods. The idea of modifying geometry in analysis codes to obtain inverse methods can be traced to the work of Barger⁵⁰. More recently, the approach has been used by Campbell⁵¹.

2.3. The Existence and Uniqueness of the Inverse Solutions

Inverse methods also require that the user have some design experience and appreciate several aerodynamic characteristics in order to specify a pressure distribution for which a geometry does exist! (aerodynamically attractive pressure distribution may require a physically unrealistic geometry or, in general, a geometry that is unattractive from the structural engineer's point of view). The existence and uniqueness problem of a inverse solution was first raised by Lighthill⁵². Using conformal mapping techniques in two dimensional incompressible flow, he demonstrated that a unique and correct solution to the inverse problem does not exist unless the prescribed pressure (or, equivalently speed) distribution satisfies certain integral constraints. Two constraints arise from the requirement that the airfoil profile have a specific trailing edge gap. A third, more subtle constraint requires that the prescribed surface speed distribution be compatible with the specified free stream speed. Woods⁵³ has pointed out that similar constraints are also required in the

mixed design problem in which the pressure distribution is prescribed for some parts of the airfoil and the shape is prescribed for other parts.

Volpe and Melnik^{54,55} have demonstrated that the role of constraints and the question of correct formulation of the inverse problem have never been properly addressed for two dimensional compressible flow and that, as a consequence, most existing inverse methods for transonic flow are not well formulated. It was pointed out that similar constraints plausibly exist for compressible flow, at least in the absence of shock waves⁵⁵. The existence of these three constraints implies that in general, the target speed distribution must contain three free parameters to guarantee that the constraints can be satisfied through proper adjustment of the parameters. Thus the originally prescribed target pressure changes in each inverse iterations to satisfy the constraints. Introducing a potential function to formulate these three constraints mathematically, they successfully applied the method to two dimensional compressible potential flow. Because particular forms are needed for the functions⁵⁶ which correct the target speed according to the constraints, it is not easy to apply the method for general two dimensional flows. But they are the pioneers who have investigated the existence of an inverse solution and formulated the constraint conditions mathematically for the compressible flow.

A similar solution existence problem was studied by Daripa⁵⁷. He concentrated on the third constraint of Lighthill's; The M_∞ obtained from the input pressure distribution using the irrotational and isentropic relations and Bernoulli's equation is compared with M_∞^c (the computed free stream Mach number which is determined by the solution of the governing equations subject to the target pressure distribution). It was pointed out that for a solution to exist, M_∞ determined directly from the input C_p and the computed M_∞^c must be the same, otherwise a solution with the input pressure distribution does not exist. The other two constraints⁵² connected to the trailing edge closure problem were not discussed.

The trailing edge closure problem was investigated by Volpe and Melnik⁵⁴, Carlson and co-workers^{58,59} and Shankar⁶⁰ *etc.* at the subsonic and transonic speeds. Prescription

of an unconstrained pressure distribution may lead to the airfoil sections which have either excessively blunt trailing edge (open) or which, at least numerically, have the upper and lower surfaces crossed at the trailing edge (fishtailed). Carlson^{58,59} used the adjustment of the nose radius (relofting) to close the trailing edge in his direct-inverse calculation. Shankar⁶⁰ altered the velocity potentials in front of the leading edge to obtain automatic trailing edge closure. Even though they ignored the role of the constraints which were suggested by Lighthill and Woods, closed airfoils and wings were generated and their methods are useful for the inverse problem in the subsonic and transonic flow.

Even though some inverse methods for the two dimensional Euler equations have been developed in the transonic speeds, the existence and uniqueness problems of the inverse solutions have not been properly resolved because of their mathematical complexities in the formulation. The inverse solution existence problem in the three dimensional cases is much worse as discussed in reference 7. The question of well-posedness does not seem to have been addressed properly even for incompressible flow⁷. One aspect is that three dimensional equivalents of Lighthill's constraints have not been formulated. Theoretical difficulties, however, do not prevent the development of useful inverse procedures⁷.

3. The Objective of the Present Study

The objective of this work is to understand the key issues for achieving low drag supersonic and hypersonic shapes, and to develop efficient axisymmetric and three dimensional supersonic and hypersonic inverse design methods using the Euler equations. In the following chapters, the minimum drag body shapes, especially for power law bodies in the supersonic and hypersonic flow regimes, will be discussed first to get a general understanding for the specification of target pressure distributions. After redefining the aerodynamically blunt and sharp bodies in the supersonic and hypersonic flows, an inverse design method for supersonic and hypersonic axisymmetric bodies will be presented. To

solve the relatively slow convergence problem in the *residual-correction type* approach, a convergence accelerating techniques will be derived and applied. Based on the axisymmetric inverse method, a new and efficient inverse method for three-dimensional bodies will be presented with several example calculations. Finally, the existence and uniqueness of inverse solution will be discussed for the developed inverse methods.

II. MINIMUM DRAG BODIES IN SUPERSONIC AND HYPERSONIC FLOW REGIMES

In this chapter, a number of classical investigations and shapes which have been found to have minimum drag under various assumptions are reviewed. Then the results of a new computational investigation are described, and new results for an optimum body shape are presented. Specific attention is paid to numerical accuracy. Results for the so-called 'hypersonic power law bodies' are presented, including a detailed examination of both the blunt nose effects, and the effects of using an equilibrium air model. A Parabolized Navier-Stokes calculation is presented to illustrate viscous effects, and finally results for one other body of interest are presented. The results of the study provide a rigorous basis upon which to draw conclusions regarding the selection of the best shapes for high supersonic/hypersonic speed vehicles.

1. Review of Some Previous Minimum Drag Studies

For circumstances where analytic theories are available and valid, minimum drag shapes have been determined theoretically for a wide range of constraint conditions. Analytic theories are available for both linearized supersonic flow and hypersonic flow when Newtonian flow theory is valid. Figure 1 presents Fink's⁶¹ summary of the situation and is based on his figure. Several results available from the analytic results for minimum drag bodies are shown. This figure also illustrates the large gap between the regions where analytic solutions are available. The gap exists in the analytic theory of minimum drag shapes for values of the Mach number similarity parameter $(M_\infty^2 - 1)^{0.5}/(l/d)$, where l/d is fineness ratio between about .6 and 4. This is the area requiring further investigation, since this is the region in which much of the flight will take place for future high speed vehicles.

The shapes considered here correspond to bodies of given length and diameter (the case that most closely corresponds to actual aircraft design requirements for current supersonic aircraft), and with their maximum cross-sectional area at the base (which has been the classical case of interest in hypersonic flow). Perhaps the most unusual feature of the classical results for both supersonic and hypersonic minimum drag bodies of given length and diameter is that the bodies have an infinite slope at the nose. If the length is unspecified a pointed tip will result⁶², and in fact be cusped. This tip is unsuitable for high speed cruise vehicles. Both structural and aerodynamic heating problems eliminate these shapes. In the case of supersonic flow, the Karman Ogive³ was derived using slender body theory. The derivation requires that the body slope at the base be zero. The resulting shape has an infinite slope but zero radius of curvature at the nose. Consistent with slender body theory, the body shape and drag exhibit no Mach number dependence. Parker⁶³ obtained the linearized supersonic theory minimum drag shape. He found that the minimum drag body was in fact pointed at the nose, but with an initial slope equal to the Mach cone angle. In his case this was the maximum slope that the analysis theory allowed. Thus the slope is large for moderate supersonic speeds for which the theory is valid. Parker's analysis did not require that the body slope at the base be zero, and a nonzero slope resulted. As shown in figure 1, his results do exhibit a Mach number dependence, predict a body with less drag than the Karman Ogive, and are valid for small values of the Mach number similarity parameter.

At hypersonic speeds Eggers *et al.*⁶² used Newtonian impact theory to show that when body length is specified, the body has a blunt nose. However, the length scale of the blunt nose is extremely small, such that solutions obtained on computational meshes appropriate for the solution over the entire body do not reveal the features of this region. Clearly the analysis in both cases demonstrates that when length is held fixed, a high pressure applied over the differential area $rdrd\theta$ is acceptable at the nose where r goes to zero. This allows reduced slopes and hence drag forces over those parts of the body where the differential area is larger. In addition, Eggers *et al.*⁶² showed that a power law body of

the form $r \sim x^n$ closely approximated the minimum drag shape obtained analytically when $n = 3/4$. Figure 2 shows the small scale of this blunt region of the body.

Figure 3 illustrates the differences between various minimum drag body shapes. Again it is clear that for the cases with blunt noses the blunt portion of the body is extremely small relative to the complete shape. Table 1 presents a summary of the various minimum drag shapes of interest here.

A slightly different shape is obtained if the so-called Busemann correction is introduced into the hypersonic flow theory. This was done by Hayes⁶⁴. In this case the shock layer is allowed to separate from the body at some distance along the surface. The drag predicted with this shape is less than the value determined by Eggers *et al.*⁶², as seen in table 1. The Hayes shape is obtained using an $n = 3/4$ power law initially, and then continuing aft of a match point, x_0 , with a shape proportional to $(x - x_0)^{1/3}$. An alternate derivation of this shape was also given by Miele⁶⁵. The minimum drag body including the Busemann correction results in a minimum drag body for which the power law form provides a very close description when $n = 2/3$. Studying the general class of power law bodies, Cole⁶⁶ also found the $n = 2/3$ power law body to be the minimum drag shape when the Busemann correction was included. A comparison of the exact Hayes' body shape with the $n = 3/4$ power law shape is contained in the results section.

Considering the Mach number (or more precisely similarity parameter) region where analytic solutions are not valid, the key contribution has been made by Fink⁶¹, who thoroughly examined, on the basis of hypersonic small-disturbance approximations to the shock-expansion method, the connection between the classical linear supersonic and hypersonic theories. In this region optimum body shapes are obtained through numerical computation. No a priori assumption was made for the body shape, and the optimum body was obtained as a numerical table of values of the radii at 25 stations along the length. His results were generally "around" the $n = 3/4$ power law body shape. Although not given explicitly, the initial vertex angle was large. In using the shock expansion method, Fink

was constrained by the theory to obtain shapes for which the initial nose angle resulted in an attached shock.

Two other results are contained in figure 1. Zandbergen⁶⁷ computed numerical solutions for optimum bodies with finite nose angles using the method of characteristics. He started his numerical calculation assuming a conical nose and then allowed the shape to seek an optimum radius distribution aft of the conical starting stations. Although he expected to obtain cusped solutions for the nose shape, the slope was always greatest along the conical segment, and began decreasing as soon as it was allowed to change. The implication is that the procedure would have led to very large initial slopes (blunt noses) if the procedure allowed for it. The other point on the plot was obtained by Powers⁶⁸, who determined optimum shapes for bodies required to have spherical noses. Powers⁶⁸ used equilibrium air blunt body and method of characteristics numerical methods and obtained the optimum shape from a class of spherical radii and perturbation polynomials to a baseline body. His optimum shape was near the 3/4 power law body, and with an extremely small spherical cap, much smaller than the one he used as the initial baseline.

In addition to the results discussed above, which were obtained some time ago, optimum bodies are still being derived using analytic theory. The recent work by Maestrello and Ting⁶⁹ used modified Newtonian theory to obtain optimum forebody shapes for the case where the leading edge radius was specified.

One other significant numerical study of shapes has been made. Stivers and Spencer⁷⁰ conducted an extensive numerical investigation of shapes. They were interested in hypersonic cruise vehicles during the surge of interest in the mid 1960's. They also used the method of characteristics, and computed the drag for four classes of bodies over the range of Mach numbers from 2 to 12. Being one of the few studies of cruise vehicles, they investigated shapes that had their maximum thickness at the midpoint. The constraint was the use of specified length and volume. The body classes studied included Sears-Haack, parabolic arc, back-to-back Karman Ogive and a Miele shape³ of given surface area and

volume rather than length. They included skin friction and base drag in their analysis. The results indicated that a Sears-Haack body with a slightly cut-off base would be the shape with the lowest total drag. The small blunt nose of the bodies was ignored in the calculation. They did not consider power law shapes.

Several experimental investigations of minimum drag bodies have been made. Initially, Eggers *et al.*⁶² conducted an experimental evaluation of their new shape. Other key investigations were conducted by Perkins *et al.*⁷¹, Spencer and Fox⁷², and Fournier, *et al.*⁷³. In each case the details of the boundary layer state are not described in the report in enough detail to undertake rigorous numerical calculation to validate numerical procedures. However, they do provide valuable relative data for a large range of flow conditions and body shapes. The experimental results confirmed the theoretical analysis.

2. Numerical Investigation

Although the advanced vehicles being considered will be three dimensional, it appears prudent to examine the minimum drag of axisymmetric bodies as a logical starting point. Following the classical supersonic aerodynamics area rule, it is expected that much of the understanding achieved from axisymmetric bodies can be carried over directly to non-axisymmetric bodies. This has been shown to be the case experimentally by Spencer and Fox⁷² at Mach number 10. As such, this means that axisymmetric body results should provide a broader basis for aerodynamic design than cases for which the details of a particular design concept have already imposed numerous constraints on the aerodynamic shape. Furthermore, it is recognized that the crucially important aero-propulsion interactions also need similar consideration. However, regardless of the aero-propulsion integration problems, the basic airframe drag reduction will still play a key role in achieving successful flight at the high speeds being considered.

In this chapter, the case for which most development has been done in the past is

addressed: the case of minimum drag for specified length and maximum area, and where the shapes have their maximum area located at the base. Power law bodies are evaluated to determine which value of the exponent results in the minimum drag when analyzed using the Euler equations. The results are presented for both perfect gas and equilibrium air thermodynamic models, over a Mach range from 3 to 12. Using the same method the Hayes' shape will be examined to determine the relative drag values. Finally, the possible viscous effects are evaluated by presenting a Parabolized Navier-Stokes calculation.

2.1 Computational Methodology and Grid Selection

A modern CFD method has been used to conduct the study. By initially considering only axisymmetric bodies and using space marching techniques, the analysis can be performed economically, allowing an in depth investigation of the aerodynamic performance of minimum drag body shapes. Such a study has been carried out using the code popularly known as **cfl3de**⁷⁴. The **cfl3de** computer code is capable of handling two dimensional, axisymmetric, and three dimensional compressible flow. The code solves the Thin Layer Navier Stokes (TLNS) equations and the Parabolized Navier Stokes (PNS) equations as well as the Euler equations with a Finite Volume formulation. Both the flux vector splitting scheme due to van Leer and flux difference splitting scheme due to Roe are incorporated in the code. Several options are available in the code i.e.

- first, second and third order spatial accuracy in i, j, k direction
- either perfect gas or equilibrium air gas model
- either space marching or global iteration technique
- many user selectable boundary conditions for $i0, idim, j0, jdim, k0, kdim$ (these are defined in Appendix A and figure A)

In this calculation the code was run using only one “slice” of the full three-dimensional grid in the circumferential direction, and enforcing the boundary condition that

there be no circumferential flow gradients, which is a conventional way for axisymmetric problems. This reduced computational time dramatically. The resulting computational grid is shown in figure 4b, compared to the usual grid system. A sample of the axial grid distribution is also included in the figure. Details of the analysis code implementation are given in Appendix A.

For the cases considered during the minimum drag body study, 41 stations were used along the body, with stations clustered near the nose. The grid consisted of stacked cross flow plane grids. For the low drag bodies studied in this chapter this approach was determined to be acceptable by comparing drag results obtained with this method against previously computed exact results. Between the body and the outer edge of the grid 20 points were used, with clustering near the body, as shown in figure 4. Typically the shock was located approximately 3/4 of the way between the body and outer edge of the grid, and, due to the grid stretching, a much higher percentage of the grid points were actually inside the shock (Details of the grid system used are given in Appendix B). Using this grid, computations were conducted on the VPI & SU IBM 3090 supercomputer. Typical CPU times were 6 minutes for the calorically perfect gas cases, and 6.7 minutes for the cases where the equilibrium air gas model was used.

2.2 Numerical Accuracy

A key advantage of the simple axisymmetric model is the ability to carry out detailed convergence studies, providing both confidence and guidance in establishing the required grid resolution for future and more costly three-dimensional calculations. An evaluation of drag results showed that the converged values were obtained by converging the solution to the level where the residual had decreased by 4 orders of magnitude, and this criteria was used for all the results presented.

The grid study was carried out to determine the grid resolution required to obtain

accurate results. The effect of grid density on results for a typical case is shown in figure 5 for drag and figure 6 for surface pressure distribution. Although the drag, as expected, is sensitive to the grid density, the 41x20 grid provides relatively accurate values while allowing a computation economical enough to be made several hundred times. The corresponding results for the pressures, shown in figure 6, clearly appear to be converged much sooner than the drag results.

3. Results for Optimum Supersonic/Hypersonic Bodies

The range of body shapes evaluated and Mach numbers used for each shape are given in table 2. The table presents drag coefficients for the perfect gas model results. In all the work presented here the drag coefficient is based on the maximum cross-sectional area.

3.1. Power Law Bodies, $l/d = 3$

Using **cf13de** for the Euler equations, a parametric study of the drag of various power law bodies was conducted for a range of Mach numbers using the calorically and thermally perfect gas model. The baseline for the study was the length to diameter (l/d) of 3 case. The Mach numbers were selected to correspond to the values at which Eggers *et al*⁶² conducted experiments. The results are presented in figure 7 as well as numerically in table 2. For comparison, the results from Newtonian flow are also included. The predicted drag is presented compared to the drag of a cone using the same method. First note that the Newtonian flow minimum occurs near $n = 3/4$ as is well known. The results from **cf13de** predict a bigger reduction in drag than the Newtonian theory, and with the minimum “ n ” occurring at the lower value of the exponent of approximately .69. Recalling that the theoretical minimum drag value of power law bodies occurs at $n = 2/3$ when the Busemann correction is included, it is seen that the numerical value falls between the pure Newtonian and the Newton-Busemann theory. Drag reductions increase with increasing

“ n ” does not vary with Mach number. Figure 7b is provided to reveal the details around the minimum drag value. The minimum is fairly flat, but for the minimum “ n ” there is a distinct and non-trivial reduction in drag compared to the $n = 3/4$ case. Finally, note that the trends predicted by Newtonian theory are very accurate for nearly conical bodies, but become poor as n becomes significantly different than unity.

Pressure distributions are presented in figure 8 for the Mach 6.28 case. The pressure distributions are all very similar for the values of “ n ” around the minimum and very different than the cone case. Note that the pressures “cross over” at approximately 20% of the length for the values of “ n ” just above and below the optimum. This demonstrates how the optimum results from a balance of higher pressures near the nose and lower pressures on the aft portion of the body.

Figure 9 presents the drag results for various key values of the power law exponent with Mach number. In this case the results are not normalized by the cone value. Experimental data obtained by Perkins *et. al.*⁷¹, and the results of an analysis of the Hayes’ body to be discussed below are also included. Despite the apparent closeness of the results for the $n = .69$, $.75$ and $.65$ bodies, recall that this scale is large, and that in the case of the F-102 aircraft, as an example, the benefits of area ruling on the order of 25-35 counts meant the difference between subsonic and supersonic flight. Thus designers should exploit the extra benefits of the $n = .69$ body. The Mach number trends and general drag levels are supported by experiment. However, experimental results are difficult to compare directly. Because of some difficulties in the variation of test Reynolds number with Mach number and related transition location uncertainties, the Eggers *et. al.*⁶² experimental data is not included in the figure. The results of Perkins *et. al.*⁷¹ included skin friction not included in the theoretical result. Without more experimental details, it appeared inappropriate to make a direct comparison between a viscous calculation and the experiment.

3.2. Power Law Bodies, $l/d = 5$

A limited analysis of the $l/d = 5$ power law body was made to determine whether the change in body slenderness had an effect on the minimum drag value of the power law exponent. Figure 10 presents the results of the calculations. It appears that there is in fact a small change in the optimum value from $n=.69$ to $n=.7$. Otherwise the trends are similar to the previous case.

3.3. Thermodynamic Gas Model Effects

At the high Mach numbers of interest, the validity of the use of a calorically perfect gas must be examined. **cf13de** was used in both the calorically perfect gas and equilibrium air models to assess the possible importance of the gas model. The results are given in table 3 for drag coefficients, and demonstrate that for low drag bodies, which minimize the disturbance to the flow, minimal effects are found. The calculation was carried out for $n=3/4$ for the inviscid analysis, and $n=.69$ for the PNS analysis. The equilibrium calculation was carried out for conditions corresponding to an altitude of about 96,000 feet.

Pressure distributions are given in figure 11. Differences between the pressure distributions obtained using the two different gas models cannot be identified. Although the pressures are nearly identical, the surface temperature is affected, as shown in figure 12. Near the nose a difference of from 50°K at $M_\infty=6.28$ to nearly 600°K at $M_\infty=12$ was found. This is not surprising based on an examination of the typical effects on pressure of various thermodynamic models for air^{75,76}.

3.4. Numerical Examination of the Nose Region

Results presented thus far ignored the explicit effect of the nose bluntness. With the

scale of the bluntness so small, see figure 2, in comparison to the rest of the geometry, normal numerical solution grids don't "see" the bluntness. An examination of the possible effect of bluntness was carried out to identify any possible effects on the conclusions obtained previously. Figure 12 provides some details of the local blunt body flow at the nose. This demonstrates that the blunt body geometry was fully accounted for in a calculation. Both perfect gas and equilibrium air results are presented.

The pressure distributions over the entire body corresponding to a global iteration solution which accounted for the blunt body and the normal space marching solution are presented in figure 13. No difference can be identified. The shock locations are shown in figure 14.

These results show that the blunt noses on minimum drag bodies are confined to such a small region that the detailed treatment of the nose is not necessary to obtain valid results for the flow over the rest of the body. The detailed phenomena of the blunt bodies near the nose region will be thoroughly investigated in the next chapter.

3.5. Effects of Viscosity

An evaluation of the possible effect of viscosity on the results was also made. Using the code in the space marching mode a PNS calculation was carried out. The computations were made for laminar flow, and the grid was changed from the nominal 41x20 grid to 51x60. The extra radial grid points were tightly spaced near the body. Figure 15 shows the results. Both equilibrium air and perfect gas models were used. Although the pressure distributions do change very slightly (about the width of the curve in the figure over a small portion of the body length), it is clear that for low drag shapes viscous effects are minor.

3.6. The Hayes Forebody Shape

3.6. The Hayes Forebody Shape

Finally, the so-called Hayes body was evaluated. Figure 16 presents his minimum drag body shape compared with the optimum power law body determined previously and the classic 3/4 body. Note that even though his body begins with a power law shape of 3/4, the value of the coefficient of x^n produces a shape that closely corresponds to the $n=.69$ power law body determined above to be optimum over the initial part of the body. Initially the Hayes' shape is larger than the other shapes before rather abruptly bending to meet the specified maximum diameter requirement. The effect of this body shape is clearly shown in the pressure distribution also shown in figure 16. As expected, the body is initially similar to the $n=.69$ case, then due to its fullness the pressure is higher than the other shapes.

The drag of the Hayes' body was included in figure 9. At $M_\infty=6.28$ for the same given length and diameter the Hayes shape has a drag 4.95% more than the $n=.69$ power law body. The same trend is observed at Mach 8. Thus the Hayes' shape appears to offer no advantage for the problem of a given length and diameter. It has more volume, and this appears to result in a higher drag.

3.7. Discussion of the Results of Optimum Body Shape

To summarize the new findings, Fink's figure is repeated as figure 17 with the results obtained in the present study included. It is clear that an analysis with an advanced code verifies the trends previously obtained, however **cf13de** predicts a drag value slightly below the values obtained previously for other shapes in this Mach number regime for the $n = .69$ power law which is between .66 and .75, previously found analytic optimum. The validity of the Fink analysis is shown for high Mach numbers. The breakdown of the HSDT assumptions is evident at the lower Mach numbers. The importance of using a complete analysis, such as **cf13de**, is clearly demonstrated.

III. AERODYNAMICALLY BLUNT AND SHARP BODIES

A detailed numerical examination for minimum drag shapes with slight bluntness at the nose leads to an improved understanding of the meaning of aerodynamically sharp and blunt shapes as opposed to geometrically blunt and sharp shapes. By defining the power law shape to be the relevant gauge function, a new criterion for the definition of an aerodynamically sharp shape will be proposed using simple Newtonian flow theory and numerical results of the power law bodies obtained in the previous chapter.

1.The Issue:A Discovery during the Analysis of Minimum Drag Shapes

What do we mean by blunt and sharp? Intuitively it seems clear. However, for preciseness a geometrically sharp body is defined to be one with a finite slope at the tip (nose or leading edge depending on whether the shape is three dimensional, typically thought of as an axisymmetric body, or two dimensional, such as an airfoil). The leading edge radius is zero, and the point feels sharp. In practice it is extremely difficult to fabricate a perfectly sharp shape. Any manufacturing error results in a significant deviation from the design contour, and furthermore sharp edges are difficult to maintain because they are easily damaged (sometimes also damaging the workers, sharp edges cause lots of injuries).

On a more fundamental basis, no shape can be made that is perfectly sharp at a molecular scale. There is always some bluntness, even on a razor blade. At what point can a shape be treated as sharp rather than blunt relative to a flowfield treated as a continuum? Practical limits to manufacturing, and the small scale of most fluid mechanics flowfields associated with bluntness effects provide us with an intuitive impression of blunt and sharp shapes which however lack precision.

The numerical results of the minimum drag body study in the previous chapter showed that at supersonic and moderate hypersonic speeds (Mach 3 to 12) low wave drag occurred for power law bodies with an exponent of approximately .7, compared to .75 or .66 for various classical hypersonic theories. A property of the power law body, and many minimum drag shapes, is the surprising geometric feature that the shape of the axisymmetric body, $r(x)$, has an initial slope, dr/dx of 90° .

During an examination of numerical solutions at the nose required to validate the integrity of the solutions obtained during the minimum drag shape study, unusual behavior was observed. It is expected to observe a classical blunt body solution at the nose ($dC_p/ds = 0$). Instead, a very different character, as shown below, was observed. This leads to further investigation of the details of the solution obtained at the nose, specifically the local geometry and related flowfield.

Thus a definition of aerodynamically as opposed to geometrically blunt and sharp shapes will be proposed, based on the improved understanding of geometry and flowfields at the nose or leading edge.

2. Geometrical Analysis for Several Minimum Drag Shapes

Consider the power law shape:

$$r = Ax^n, \quad (3.1)$$

and the formula for radius of curvature:

$$R(x) = \frac{[1 + (dr/dx)^2]^{3/2}}{\left| \frac{d^2r}{dx^2} \right|}. \quad (3.2)$$

For the power law body given by equation (3.1) the value of the radius of curvature $R(x)$ becomes:

$$R(x) = \frac{I}{|n(n-1)A|} \left[x^{\frac{2}{3}(2-n)} + (nA)^2 x^{\left(\frac{4}{3}n - \frac{2}{3}\right)} \right]^{\frac{3}{2}} \quad (3.3)$$

where the value of the nose radius is found by taking the limit of the equation as $x \rightarrow 0$. The first term in the square bracket will vanish unless $n \geq 2$. The exponent of the second term will control the result in practical cases. The result is:

n	$R(0)$
$> 1/2$	0
$= 1/2$	$A^2/2$
$< 1/2$	∞

Note that traditional airfoils are in fact parabolic near the nose, the leading term in the definition of the shape is $x^{1/2}$, and thus they have finite values of the leading edge radius. The power law bodies that have minimum drag, $n = .7$, in fact have a zero leading edge radius, even though they are geometrically blunt.

The identification of a class of shapes that do not have a nose radius, yet have an initial slope of 90° is difficult to visualize. Figure 18 shows as an example the nose region of the $3/4$ power law body with a circle inscribed tangent to the 45° slope point on the power law body. The result of zero leading edge radius means that as we try to get closer and closer to the nose with increasingly smaller circles the actual body always “sticks out,” as occurs in this figure. Figure 19 provides examples of typical curvature distribution near the nose for three power law bodies representing the three different cases of limiting behavior.

Having conducted an analysis for the hypersonic case, the classical supersonic cases of the Karman Ogive and Sears-Haack Body were examined.

- Karman Ogive⁷⁷

$$S = \pi r^2 = \frac{S(l)}{\pi} \left[\pi - \cos^{-1} \left(2 \frac{x}{l} - 1 \right) + 2 \left(2 \frac{x}{l} - 1 \right) \sqrt{\frac{x}{l} \left(1 - \frac{x}{l} \right)} \right] \quad (3.4)$$

$$R(x) = 2\sqrt{\pi} \frac{\left[S + \frac{l}{4\pi} \left(\frac{dS}{dx} \right)^2 \right]^{\frac{3}{2}}}{\left[S \left(\frac{d^2S}{dx^2} \right) - \frac{l}{2} \left(\frac{dS}{dx} \right)^2 \right]} \quad (3.5)$$

where, S is a cross sectional area.

- Sears-Haack Body⁷⁸

$$r/l = \frac{1}{2f} (1 - \zeta^2)^{\frac{3}{4}} \quad (3.6)$$

$$R(x) = \frac{2f}{3} \frac{(1 - \zeta^2)^{\frac{1}{2}}}{2 - \zeta^2} \left[(1 - \zeta^2)^{\frac{1}{2}} + \frac{9\zeta^2}{4f^2} \right]^{\frac{3}{2}} \quad (3.7)$$

where, f is a fineness ratio related to the length and volume (V) by,

$$f = \sqrt{\frac{3\pi^2 l^3}{64 V}} \quad \text{and} \quad \zeta = 1 - 2 \frac{x}{l}$$

In each case it was found that the leading edge radius was also zero. Once this was discovered, the question arises as to how these shapes behave near the leading edge or nose. The behavior of the shapes was investigated by determining the proper gauge

function, such that the ratio of the shape to the gauge function remained finite as $x \rightarrow 0$. The power law body was picked as a candidate gauge function, with an arbitrary value of the power law exponent, n . The results of the analysis are that both the Karman Ogive and Sears-Haack bodies behave near the nose as a power law body with $n = 3/4$, the same value as one of the theoretically derived minimum drag cases at hypersonic speeds².

$$\begin{aligned} \text{When } n = 3/4, r_{K-O} &= O(r_{\text{power law body}}) \\ r_{S-H} &= O(r_{\text{power law body}}) \quad \text{as } x \rightarrow 0. \end{aligned}$$

Further analysis confirms that not only do the Karman Ogive and Sears-Haack bodies behave similarly to the $3/4$ power law body, but that the radius of curvature distribution near the nose is described by the $3/4$ power law body gauge function also.

$$\begin{aligned} \text{When } n = 3/4, R_{K-O} &= O(R_{\text{power law body}}) \\ R_{S-H} &= O(R_{\text{power law body}}) \quad \text{as } x \rightarrow 0. \end{aligned}$$

3. Newtonian Flow/cfl3de Analysis

The motivation for the investigation originated in the examination of the numerical solution for the power law bodies described in the previous chapter in the vicinity of the nose. It is expected to see classical blunt body pressure distributions, with the initial pressure gradient, $dC_p/ds = 0$. Instead, solutions with very large initial pressure gradients were obtained. Because of the interest in accurate solutions for drag this effect was examined closely, with numerous grid refinements and using a variety of different (but equivalent) boundary conditions and numerical solution schemes available in the code (cfl3de⁷⁴). In addition to close examination of the numerical results, an exact solution for the pressure distribution was obtained using Newtonian flow theory. For several values of power law index n , Newtonian solutions and numerical results for the pressure coefficient are shown in figure 20 and 21. Big differences in C_p distribution are found between power

law bodies of $n > 0.5$ and $n < 0.5$.

Assuming Newtonian flow and considering the power law shape of $r = Ax^n$, the pressure coefficient at the body surface is given by

$$C_p = 2\sin^2 \theta \quad (3.8)$$

$$\text{where, } \theta = \tan^{-1} \left(\frac{dr}{dx} \right) \quad (3.9)$$

The pressure gradient along the arc length from the stagnation point,

$$\frac{dC_p}{ds} = - \left[\frac{4(1-n)}{A^3 n^3} \right] \frac{x^{(2-3n)}}{\left[1 + A^{-2} n^{-2} x^{2(1-n)} \right]^{\frac{5}{2}}} \quad (3.10)$$

Thus,

n	dC_p/ds
$> 2/3$	$-\infty$
$= 2/3$	$-4(1-n)/A^3 n^3$
$< 2/3$	0 as $x \rightarrow 0$.

The results are essentially the same as the numerical results, confirming that the initial pressure gradient for the flow is not zero, and quite large. In figure 22, the pressure gradients are shown as a function of arc length for several power law indices n . In fact, the Newtonian solution results in an infinite pressure gradient exactly at the nose. The singularity is $O(s^{(2-3n)/n})$. For $n = 3/4$, the singularity is $O(s^{-1/3})$ for dC_p/ds , where s is the arc length from the stagnation point. At several locations on the body, the pressure gradients are shown as a function of power law index n in figure 23. For $n < 2/3$, dC_p/ds is nearly zero near the nose.

4. Implications for Nose Shaping

The fundamental advantage in using a blunt body is the substantial decrease of wave drag and significant reduction of heat flux from the heated gas to the surface (a significant amount of heat is dissipated in the form of losses in the bow shock wave)⁷⁹. The result described above provides an unfortunate implication for design work for bodies at the higher Mach numbers. Although the nose is geometrically blunt for low drag bodies, the heat transfer will still be large, because they are aerodynamically sharp. If the material cannot withstand the heating environment then the aerodynamically sharp body must be abandoned in favor of a body with an identifiable nose radius to control the heat transfer. This problem was examined recently by Maestrello and Ting⁶⁹.

The present analysis provides a new definition for the aerodynamic sharpness and bluntness using the power law shape as a gauge function: An aerodynamically sharp shape is a shape that is $O(x^n)$, where $n > 1/2$. Thus, the minimum drag power law shapes have low drag and behave as though they are sharp even though they are not geometrically sharp. With a general understanding for the surface pressure distributions of the supersonic and hypersonic minimum drag bodies investigated in the present and previous chapters, an axisymmetric inverse Euler method will be developed in the next chapter.

IV. INVERSE DESIGN METHOD FOR AXISYMMETRIC BODIES

In this chapter, a new inverse method using the Euler equations is developed for supersonic and hypersonic axisymmetric bodies. The emphasis is placed on obtaining an efficient and robust method throughout a wide range of supersonic and hypersonic speeds. For bodies which have very slightly blunted noses, previous work (Chapter II and III) has demonstrated that space marching can be used even though there exists a detached shock and locally subsonic region just at the nose. The round nose induced subsonic axial flow region will occur on a scale much smaller than the aerodynamic design length scale for atmospheric cruise vehicles. Thus the effect of this region can be neglected and a space-marching procedure be used to minimize computing time when a body shape is designed for prescribed surface pressure distributions. For cases where this region must be explicitly accounted for, the present method can be used downstream of the embedded subsonic region.

After a general description of the method, an analysis of numerous rules relating local geometry changes to pressure changes is presented. Results of numerical experiments are then presented to select the rule adopted in the new method. Then, several examples are presented to demonstrate the performance of the method.

1. Description of the Method

1.1. General Approach

Following reference 32, an existing analysis method is used as the basis for the inverse procedure. Minimum of modifications are made, and the modifications for easy

implementation in new analysis methods is to be made as they become available. For the computations presented here the code **cf13de** is used again, which was introduced in the minimum drag body study. The code is capable of handling general three-dimensional subsonic, supersonic and hypersonic flows, although the current inverse method is applied to axisymmetric bodies at zero angle of attack. This allows the method to be developed with a minimum of computational expense. The inverse method developed in this chapter is directed toward design at supersonic and hypersonic speeds, and the space marching option of **cf13de** is used for economy.

The details of the modification to the baseline analysis code are shown in the flow chart in figure 24. The method steps one grid plane at a time, using the same approach at each station. One initial plane of data, including pressure and geometry is required to start the calculation. The key item is the surface pressure-body shape rule for changing the shape from the initial estimated value to the value required to obtain the desired pressure. This is an iterative procedure, and the objective of this work is to minimize the number of iterations required to obtain the design pressure. In fact, the analysis of candidate rules described below is a key contribution of the inverse procedure. Thus, the required information is:

- design free stream Mach number and nominal body geometry,
- starting data plane for the inverse calculation,
- target pressure distribution.

The mechanics of the process are as follows. On the i -th plane, first determine whether we use the inverse routine or not. If not, simply obtain the flow properties using the direct analysis code and then go to the next plane ($i+1$ st plane). When we use the inverse routine, we specify the target pressure at the present (i -th) plane and assume the initial surface pressure P_i by,

- directly, surface pressure at the i -th plane equals the target pressure at that plane or,
- extrapolating pressure at i -th plane with pressures at previous planes using Lagrangian polynomials

Calculations demonstrated that the first choice is better (it has a slightly faster convergence behavior). Given the initial pressure P_i , the local Mach number M_L , at the i -th plane is calculated from the point isentropic relations. This local Mach number is then used in a local application of surface pressure-body shape rule.

During the iterations within the inverse routine, M_L is updated with the calculated pressure (M_L can be fixed during the inverse iterations but convergence is worse). Using one of the surface pressure-geometry relations described in the next section, the geometry at iteration k is obtained from the calculated pressure (P_i) and local Mach number (M_L) using the equation

$$\left(\frac{dz}{dx}\right)^k = \left(\frac{dz}{dx}\right)^{k-1} + \frac{d}{dC_p} \left(\frac{dz}{dx}\right)^{k-1} (C_p^T(i) - C_p^{k-1}(i)) \quad (4.1)$$

where C_p^T is the target pressure coefficient. The pressure and body slope are computed here at the center of the cell, station $i-1/2$, consistent with the finite volume numerical scheme.

The body shape at station i is then constructed assuming a constant slope between the i and $i-1$ computational stations. The body slope is second order accurate since it is found from a central difference. The resulting shape is formally first order accurate. However, computations have demonstrated that accurate results are obtained using the method.

The new grid system is generated next from the new geometry, and the computation is performed with this new geometry. The inverse iteration procedure proceeds until the difference of the calculated pressure and target pressure is within some prescribed tolerance. In most cases, convergence criterion, δ of order 10^{-4} is acceptable, i.e.,

$$|C_p^{(k)}(i) - C_p^T(i)| \leq \delta \quad (4.2)$$

For these calculations, the grid is defined at each station by stacking constant x -plane grids, and simply re-defining a simple z -grid from the surface to a point outside the bow shock wave.

In many subsonic and transonic inverse methods previously developed, a starting body shape was required to start the inverse calculation. By using space-marching for supersonic and hypersonic flow calculations, a starting body shape is not required. In each marching plane, the body shape is determined from the initial surface pressure using one of the pressure-shape rules. Moreover, we are not concerned with the existence of the resulting body a priori because the body shape is found in each marching plane, and the solution can simply be halted automatically if the specified pressure distribution leads to a body that would require a negative radius according to the calculation. This is a significant advantage in the present space-marching inverse Euler method. There will be a detailed discussion for the inverse solution existence and uniqueness in the later chapter.

1.2. Surface Pressure and Body Geometry Relations

It is important to select a good approximation for the surface pressure to body geometry relation. This is a key consideration for rapid convergence and low computational time. However, this surface pressure to body geometry approximation need not be extremely accurate since convergence will be attained through a series of iterations. It may even be quite crude, but should be robust enough to guarantee convergence, and thus must generate a meaningful body geometry for a wide range of target pressure distributions³³.

1.2.1. Several Surface Pressure-Body Shape Rules

Many approximations of the surface pressure to the body geometry can be identified. Among these, approximations which are relatively easy to apply and appropriate to the

speed range have been considered as candidates for the axisymmetric inverse calculation procedure. The surface pressure-body geometry rules which were considered are:

- (1) Linearized supersonic theory⁸⁰

$$C_p = \frac{2}{\sqrt{M_\infty^2 - 1}} \tan \theta \quad (4.3)$$

- (2) Two-dimensional shock-expansion formula which is valid from supersonic to moderate hypersonic flow⁸¹

$$C_p = \frac{2}{M_\infty} m \sin \theta + \frac{\gamma + 1}{2} (m \sin \theta)^2 + \left(\frac{\gamma + 1}{4} \right)^2 M_\infty (m \sin \theta)^3 \quad (4.4)$$

$$\text{where, } m^2 = \frac{M_\infty^2}{M_\infty^2 - 1} \quad (4.5)$$

- (3) Tangent cone approximation in hypersonic flow⁷⁹

$$C_p = \tau \frac{k_I^2}{M_\infty^2} \left(1 + \frac{1}{5\gamma k_I^{\frac{2}{3}}} \right) \quad (4.6)$$

$$\text{where, } k_I = M_\infty \theta \text{ and} \quad (4.7)$$

$$\tau = \frac{2(\gamma + 1)(\gamma + 7)}{(\gamma + 3)^2} \quad (4.8)$$

- (4) Newton impact theory⁸²

$$C_p = 2 \sin^2 \theta \quad (4.9)$$

- (5) Conical flow in supersonic region (empirical formula)⁷⁹

$$C_p = \left(0.0016 + \frac{0.002}{M_\infty^2} \right) \theta^{1.7} \quad (4.10)$$

- (6) HABP Mark III tangent cone empirical method⁸³

$$C_p = \frac{48 M_{ns}^2 \sin^2 \theta}{23 M_{ns}^2 - 5} \quad (4.11)$$

$$\text{where, } M_{ns} = 1.090909 M_\infty \sin \theta + \exp(-1.090909 M_\infty \sin \theta) \quad (4.12)$$

(7) Edwards tangent cone empirical method⁸³

$$C_p = \frac{48 M_{ns}^2 \sin^2 \theta}{23 M_{ns}^2 - 5} \quad (4.13)$$

$$\text{where, } M_{ns} = (0.87 M_\infty - 0.544) \sin \theta + 0.53 \quad (4.14)$$

(8) Combination of second order slender body theory and approximate cone solution of Hammitt and Murthy⁸³

$$C_p = \left(\frac{P_\infty}{\frac{1}{2} \rho_\infty V_\infty^2} \right) \left[\left(\frac{2\gamma}{\gamma+1} M_\infty^2 \sin^2 \theta_s - \frac{\gamma-1}{\gamma+1} \right) \left(1 + \frac{\gamma M_\infty (\theta_s - \theta)^2 \cos^2 \theta_s}{1 + \left(\frac{\gamma-1}{2} \right) M_\infty^2 \sin^2 \theta_s} \right)^{-1} \right] \quad (4.15)$$

Using these pressure-body geometry rules, the inverse routine is applied to example calculations and the most promising pressure-body geometry rule, that is, the one which has the most robust convergence behavior and least computation time is identified. In the process a standard numerical method, which appears to be unique in its application to inverse calculations, will be demonstrated and compared with the other methods. To use the rules given above a different form is required. Define a coefficient, $A(M_L, \theta)$, relating the surface pressure change to the body geometry change for each pressure-geometry rule. This coefficient should properly reflect the change of the surface pressure on the corresponding change of the body geometry. Otherwise, the resulting body geometry oscillates or diverges during the inverse iterations. The rules given previously result in a wide range of values for the coefficient. Convergence depends on the numerical value of this coefficient, which is a function of the local Mach number and the body shape.

Of these eight surface pressure-body geometry rules, rules (5) through (8) will not be further considered. Rule (5) was rejected because this rule would not produce a converged body geometry: the coefficient $A(M_L, \theta)$ between the surface pressure and body geometry is too small. Rule (6) and (7) need another approximation to M_{ns} (Mach number normal to the shock) and (8) has an unknown variable, θ_s (shock angle), so these rules are not appropriate for the inverse calculation. Thus, only the four rules (1) through (4) are considered. Henceforth, rules (1) through (4) are called as the linearized supersonic, shock-expansion, tangent-cone and Newtonian pressure-shape rules respectively.

1.2.2. ΔC_p vs. $\Delta \theta$

Equation (4.1) can be equivalently expressed using a body slope angle, θ , as follows,

$$\Delta C_p = A(M_L, \theta) \Delta \theta, \quad (4.16)$$

where, ΔC_p is $C_p^T - C_p^{k-1}(i)$ and $\Delta \theta$ is $\arctan(dz/dx)^k - \arctan(dz/dx)^{k-1}$, for use as given in equation (4.1). We can take M_∞ as the local Mach number, M_L following reference 32. $A(M_L, \theta)$ is defined as the change of surface pressure with respect to the change of body slope angle. This is determined by the surface pressure-body shape rule given above in equations (4.3) through (4.9). Using this form, the pressure-geometry rules become:

(1') Linearized supersonic pressure-shape rule

$$A(M_L, \theta) = \frac{2}{\sqrt{M_L^2 - 1}} \sec^2 \theta \quad (4.17)$$

(2') Shock-expansion pressure-shape rule

$$A(M_L, \theta) = m \cos \theta \cdot \left(\frac{2}{M_L} + (\gamma + 1)m \sin \theta + \frac{3}{16}(\gamma + 1)^2 M_L m^2 \sin^2 \theta \right) \quad (4.18)$$

$$\text{where, } m^2 = \frac{M_L^2}{M_L^2 - 1}. \quad (4.19)$$

(3') Tangent cone pressure-shape rule

$$A(M_L, \theta) = \tau \left(2\theta + \frac{1}{2}c\theta^{-\frac{1}{2}} \right) \quad (4.20)$$

$$\text{where, } c = \frac{1}{5\gamma M_L^{\frac{3}{2}}} \quad (4.21)$$

(4') Newtonian pressure-shape rule

$$A(M_L, \theta) = 2\sin 2\theta \quad (4.22)$$

$A(M_L, \theta)$ is shown as a function of the body surface slope, θ in figures 25 and 26 for two different Mach numbers (3.0 and 6.0) for each rule. The value of $A(M_L, \theta)$ must be positive because ΔC_p and $\Delta\theta$ must have the same sign for all slopes θ (both compression and expansion).

The value of $A(M_L, \theta)$ for the shock expansion pressure-shape rule is positive for the whole body slope range and changes properly with θ . Thus, this rule can be applied to any target pressure distributions. It is essentially exact for two dimensional flow.

$A(M_L, \theta)$ for the linearized supersonic pressure-shape rule is nearly constant for a wide range of body slope. With the shock expansion result being nearly correct, it is seen that the accuracy of the linearized supersonic rule is poor. Using this rule either the converged body geometry could not be obtained (oscillating during the inverse iteration for a hypersonic Mach number, say $M_\infty = 6.28$) or, took too much time to obtain a converged solution (at supersonic speeds). This pressure-shape rule, which was proposed by Barger and Brooks⁵⁰ to apply to the inverse method, is difficult to apply to hypersonic flows without modification because it fails to capture the hypersonic flow physics, resulting in convergence problems.

The tangent cone and Newtonian pressure-shape rules assume positive surface pressures, and thus these rules have convergence problems for regions of negative pressure

specification. The value of $A(M_L, \theta)$ in these rules is negative for the negative body slope region as can be seen in figures 25 and 26, and a converged body shape cannot be obtained with negative values of $A(M_L, \theta)$ as described above. This means that these rules cannot be applied to the regions of negative pressure coefficient specification. Moreover, the value of $A(M_L, \theta)$ for the tangent cone pressure-shape rule becomes very large (infinite for the particular approximation used here as the body slope vanishes). This means that a converged body shape cannot be obtained from the given pressure specification in negative or zero body slope regions.

1.3. Adaptation to Incorporate Root Finding Scheme

The convergence issues discussed above can be resolved by treating the iteration for the body shape corresponding to the prescribed pressure at a particular station as a root finding problem. This procedure will be illustrated in the next section.

Based on initial results using the basic method, an alternate scheme was adopted. Mathematically, the objective of the inverse method is to find the body geometry which will make the difference between the pressure at the current iteration and the target pressure zero. Thus, the inverse routine can be considered a root-finding scheme between ΔC_p and body slope (the body geometry is updated during the inverse iterations to drive ΔC_p to zero). The regula-falsi root finding scheme (a bracketing root finding method which is very robust in finding the root) was incorporated into the inverse method. At least two inverse iterations using equation (4.16) are needed to start the regula-falsi scheme. The first two values which are obtained from the normal operation of the inverse procedure have great influence on the convergence behavior as is the case of most root finding schemes. Thus, the convergence behavior is quite different in each pressure-shape rule because different pressure-shape rules predict different starting values. After its introduction in these cases it was adopted as part of the general method. The result of introducing the regula-falsi scheme into the method will be illustrated in the results section.

2. Numerical Evaluation for Axisymmetric Bodies

The inverse routine and pressure-shape rules which were developed and explained in the previous sections, are now applied to supersonic and hypersonic axisymmetric body test cases.

The primary objectives of the example calculations are:

- To validate the developed inverse code procedure,
- To find the most robust pressure-shape rule for the considered flow regimes,
- To clarify the convergence and the computation time for each surface pressure-body shape rule,
- To demonstrate the capability of the inverse scheme, which was developed for arbitrary body shapes including bodies which have extreme surface pressure distributions (these include pressures near vacuum pressure) and for a body with a corner,
- To understand the exact connection of surface pressure to body shape. Hence to predict the body shape from the specified surface pressure distributions.

Based on these objectives, inverse calculations were performed for six different target pressure distributions in the supersonic and moderate hypersonic speed range. Four approximations of the surface pressure to the body shape and supersonic and hypersonic Mach number effects are investigated for each case. Using this information, the method which has the fastest convergence (least computation time), and demonstrates the most robust behavior was identified.

For these calculations 41 stations were used along the body, with stations clustered near the nose. Between the body and outer edge of the grid 20 points were used, with clustering near the body. The basic grid structure is the same as that of minimum drag body calculation in Chapter II. The inverse calculation starts arbitrarily at 10% of the length from the nose with the initial body being a $3/4$ power law shape of fineness ratio 3, which was

investigated over a wide Mach number range (from $M_\infty = 3.0 \sim 12.0$) in Chapter II.

The axial computational grid at $M_\infty = 6.28$ is shown in figure 27 for the nominal body (3/4 power law shape). Figures 27b and 27c illustrate the grid arising for two inverse calculations. Typical CPU time for the first 10% of the length from the nose (marching planes $i = 1 \sim 13$) is 3 minutes. It takes 53 seconds to compute the flow over the rest of the body (from $i = 14 \sim 41$) in the analysis calculation. The CPU time for inverse calculations is heavily dependent on the pressure-shape rules, and will be examined in detail below.

2.1. Hypersonic Minimum Drag Body (Power Law Shape)

To validate the inverse calculation routine developed for the axisymmetric supersonic and hypersonic bodies, sample calculations are performed for the 3/4 power law shape calculated in Chapter II. The inverse calculations are performed for two free stream Mach number, 3.0 and 6.28 by specifying the known pressure distribution and checking the predicted body shape with the actual analytic shape. The results showed that the known body geometry could be found typically within 2 inverse iterations at each marching station by using tangent cone pressure-shape rule. The CPU time is approximately twice that of the direct calculation. Having verified the basic accuracy of this inverse method, several test cases were undertaken.

2.2. Test Cases Having Quadratically Varying Target Pressures

Four test case target pressure distributions have been selected and are shown in figure 28. Each one is discussed below.

2.2.1. Test case I & II

Test target pressure I & II are selected to demonstrate the capability to obtain the body geometry for negative pressure coefficient distributions (including pressures near vacuum: $C_{p_{vac}}$ is -0.036 for $M_\infty = 6.28$). The two target pressure distributions are prescribed as follows,

- Test case I : Pressure is minimum (near $C_{p_{vac}}$) at 50 % of the body length from the nose and then increases quadratically to the end of the body
- Test case II: Specified pressure decreases continuously from the starting point of the inverse option to the end of the body where the pressure is minimum (near $C_{p_{vac}}$ for $M_\infty = 6.28$)

As mentioned in the previous section, using the linearized supersonic pressure-shape rule, a converged body geometry cannot be obtained from the given target pressure for $M_\infty = 6.28$ (hypersonic flow), but for $M_\infty = 3.0$, the value of $A(M_L, \theta)$ is larger than that for $M_\infty = 6.28$ and is large enough to get a converged solution. However, this pressure-shape rule requires relatively long computational times to get a converged solution. Typically, the CPU time is 4 ~ 5 times longer than that of the direct calculation and the number of inverse iterations in each marching plane is 1 ~ 6. These findings are summarized in tables 4 and 5. Since this pressure-shape rule comes from the linearized supersonic theory approximation, it is not surprising that it does not work for higher Mach numbers (in this case, $M_\infty = 6.28$).

Based on this initial experience, the variation in the scheme was adopted. The effect of introducing regula-falsi scheme is clearly shown in table 4 and 5. Using this method, a converged geometry can be obtained for the given surface pressure distributions, a converged geometry is obtained for $M_\infty = 6.28$ and moreover, the convergence itself is improved for $M_\infty = 3.0$ (approximately 1.2 times faster for test case I and II). It also requires a smaller number of inverse iterations in each space marching plane than that without regula-falsi scheme (it has 1 ~ 3 inverse iterations in each plane).

The shock expansion pressure-shape rule comes from the shock-expansion theory for supersonic and hypersonic flow. Thus, it is applicable to the regions of negative body slope and negative pressure coefficient distributions. Eggers and Savin⁸⁴ noted that hypersonic flow over three-dimensional bodies can be approximated as locally two-dimensional and hence, two-dimensional shock-expansion theory is a valid approximation. In other words, this approximation, in nature, has wide applications (this can be expected from the fact that the value of $A(M_L, \theta)$ is greater than zero for all body slope θ in figures 25 and 26). As can be seen in the table 4 and 5, converged solutions were obtained for all the test cases. The only problem is that it requires relatively high computation time and a larger number of inverse iterations in each marching plane. Typical CPU time is 4 ~ 6 times longer than that of the direct calculation for test case I and 3 ~ 5 times longer for test case II and the number of inverse iterations in each marching plane is 1 ~ 10 for test case I, 1 ~ 5 for test case II, respectively. With the application of the regula-falsi root finding scheme as found in the linearized supersonic pressure-shape rule, CPU time reduces greatly (1.2 ~ 1.5 times faster for test case I and test case II)

Because of the negative pressure coefficient regions in test case I and II, the resulting body geometry will have negative and zero slopes at some stations. As shown in the previous section, the tangent cone and Newtonian pressure-shape rules have difficulties near zero and negative body slope. $A(M_L, \theta)$ becomes very large for the tangent cone rule and it goes to zero in the Newtonian rule near zero slope. Neither of these situations is suitable for inverse calculation. Furthermore, the values of $A(M_L, \theta)$ in these rules are negative when the body slope is negative. As mentioned earlier, a converged body geometry cannot be obtained with the negative value of $A(M_L, \theta)$. In an attempt to apply this rule to the arbitrary pressure distributions, the equations in (3') and (4') were modified in an ad hoc manner as follows,

$$A(M_L, \theta) = \tau \left(2\theta + \frac{1}{2} c |\theta|^{-\frac{1}{2}} \right). \quad (4.20 \text{ a})$$

$$\text{and, } A(M_L, \theta) = 2 |\sin 2\theta|. \quad (4.22 \text{ a})$$

These modifications were used with the regula -falsi scheme again in order to get a converged solution for target pressure distributions containing negative pressure coefficient regions. Using the regula-falsi scheme, the convergence rate, number of inverse iterations in each marching plane and total CPU time are greatly improved, as shown in tables 4 and 5. The modified tangent cone rule has the fastest convergence and least computation time among any other pressure-shape rules considered (about 2.0 ~ 2.9 times longer than that of the direct calculations). A converged solution is obtained within 2 ~ 3 inverse iterations in each marching plane. This is remarkable comparing other inverse codes developed previously, for which more than 10 inverse iterations are typically required³³.

The computation time for $M_\infty = 6.28$ is slightly less than that for $M_\infty = 3.0$ even though there is no difference in computation time between $M_\infty = 6.28$ and 3.0 for the direct calculation. That's because the tangent cone pressure-shape rule was derived for hypersonic flow.

The number of inverse iterations and the ratio of CPU time of inverse calculations to that of the direct calculations for all four pressure-shape rules with and without regula-falsi root finding scheme are given in tables 4 and 5. The calculated body geometries for the given target pressure distributions are shown in figures 29 and 30 for Mach numbers of 3.0 and 6.28. The calculated body for $M_\infty = 6.28$ is more sensitive to the target pressure than that for $M_\infty = 3.0$. This is because the body slope is roughly proportional to local Mach number for the given pressure coefficient.

2.2.2. Test case III

The test target pressure III was shown in figure 28 with the other target pressure distributions. It decreases slowly to 40% of the body length from the nose and then increases quadratically to the end of the body. When the target pressure distributions do not

have negative pressure coefficients, the body shape will have a positive slope over the whole region, i.e. θ will be greater than zero everywhere. Thus convergence was achieved for all the pressure-shape rules without using regula-falsi except for the linearized supersonic pressure-shape rule at $M_\infty = 6.28$.

The convergence behavior and number of inverse iterations required to obtain the solution are tabulated in table 6 for all pressure-shape rules with and without the regula-falsi scheme. The calculated body geometries for the given target pressure distribution are shown in figure 31 for Mach number 3.0 and 6.28. As expected from the positive target pressure coefficients, the body diameter increases from the nose to the end of the body. The abrupt increase of the body thickness in the end region will cause a large drag increase.

Although a converged body geometry could not be obtained using the linearized supersonic pressure-shape rule directly, by using the regula-falsi scheme a converged solution is obtained and its convergence rate is improved 1.66 times at $M_\infty = 3.0$ (typical numbers of inverse iterations are 5 and 3 for this pressure-body shape rule with and without regula-falsi scheme, respectively).

For the shock expansion pressure-shape rule the convergence rate, number of inverse iterations required, and computation time characteristics are not particularly good (only slightly improved compared to the linearized supersonic rule), although the rule is robust enough to get a converged solutions for any well-posed target pressure distribution (typical number of inverse iterations is 2 ~ 5). With the regula-falsi scheme, the computation time reduces 1.36 times for $M_\infty = 3.0$ but shows little difference at $M_\infty = 6.28$. The CPU time is 3 ~ 4 times longer for $M_\infty = 6.28$ and 4 ~ 6 times longer for $M_\infty = 3.0$ than that of direct calculation.

The tangent cone pressure-shape rule demonstrates rapid convergence and a small number of inverse iterations to obtain a converged solution. The converged solution can be obtained within 2 iterations for all the marching planes. So no improvement can be found

by the application of regula-falsi scheme. The CPU time is less than twice that of the direct analysis calculation for $M_\infty = 6.28$ and 2.5 times longer for $M_\infty = 3.0$. This is very similar to the results found for test cases I and II.

The Newtonian pressure-shape rule results in rapid convergence and a small number of iterations for $M_\infty = 6.28$ (3 times longer than that of the direct calculation) but has slow convergence behavior for $M_\infty = 3.0$. This is because the Newtonian rule reflects the actual physics of the flowfield at high Mach number. This rule is not appropriate to lower supersonic Mach numbers. It also requires positive surface pressure coefficients and has the same limitations as the tangent cone pressure-shape rule.

2.3. Test Case IV

Test case IV is chosen to demonstrate the method's ability to obtain the body geometry resulting from an abrupt change of the surface pressure distribution. The test target pressure coefficient distribution is selected to decrease linearly from the starting plane of the inverse calculation and then is held at zero when that value is reached (30 % of the body length from the nose to the end of the body).

The target pressure distribution and the calculated body shape for each Mach number is given in figure 32. From this figure, the mild corner is seen in the body geometry in the region where the specified pressure changes abruptly to zero. Note that the body radius continues to increase even though the pressure coefficient is zero. This test case shows that the method can handle the case of a target pressure distribution with an abrupt change without difficulty.

The tangent cone pressure-shape rule with regula-falsi scheme is applied for $M_\infty = 3.0$. The number of inverse iterations in each space marching plane is 0 ~ 3 (maximum number of inverse iterations occurred at the corner) and the total CPU time is 2.3 times

longer than that of a direct calculation. The linearized supersonic pressure-shape rule with the regula-falsi scheme is applied for $M_\infty = 6.28$. The number of inverse iterations in each space marching plane is 2 ~ 3 (maximum number of inverse iterations occurred at the corner) and total CPU time is 3.6 times longer than that of direct calculation.

2.4. Ogive Cylinder (Body with a Corner, $l/d = 3.0$)

To demonstrate the capability of obtaining a body shape with a corner, a known geometry (secant-ogive cylinder) was selected as a test case. An ogive has the advantage of providing without unduly increased drag a large apex angle and greater nose volume than a cone of the same fineness ratio. The planform shape of an ogive nose is defined by the arcs of a circle, and secant ogives are completely defined by the criteria,

- the radius of the ogive contour or arc in terms of the body diameter, r_s/d
- the ratio of the radius of the ogival contour and the radius of the tangent ogive, r_s/r_t

To consider a body with a corner, a secant ogive with cylindrical afterbody which has a fineness ratio of 3 was selected (fineness ratio $l/d = 3.0$ is selected to compare the drag with the power law shape calculated in the previous chapter). The corner is located at 2/3 of the body length from the nose and the secant ogive of $r_s/r_t = 2.0$ is selected because it is known that it has the smallest drag⁸⁵. The body shape of the ogive cylinder considered and the target pressure distributions for each free stream Mach number are shown in figure 33.

The target pressure distributions for free stream Mach numbers of 3.0 and 6.28 were obtained by running the direct analysis code for this ogive cylinder. The outer grid boundary was selected to be conical (the shock shape for the secant ogive was found to be similar to that of a cone), and adjusted to locate the shock approximately 3/4 of the way between the body and outer edge of the grid. Using the pressure distribution obtained from the direct calculations, the inverse calculations were made using the tangent cone pressure-

shape rule with regula-falsi. The starting plane of inverse options is 10% of the body length from the nose as before.

The results for this case were obtained without difficulty, exactly duplicating the body shape shown in figure 33. The abrupt change of the pressure at the expansion corner of the body and the difference of the pressure for each different Mach numbers can be seen. The 3/4 power law body shape and its pressure distribution are also included for the comparison. The pressure expansion at the corner caused no problem. This demonstrates that this inverse method is very robust and is also extremely efficient. For $M_\infty = 6.28$, the number of inverse iterations in each marching plane is 0 ~ 1 except near the corners (4 ~ 6 iterations). The total CPU time is 1.8 times (less than 2' indicates an extremely efficient algorithm) longer than that of the direct calculation. For $M_\infty = 3.0$, the number of inverse iterations in each marching plane is 0 ~ 1 except near the corner (3 ~ 6 iterations) and the total CPU time is 1.75 times longer than that of the direct calculation.

Although of indirect interest for the actual inverse method demonstration, the resulting drag obtained for the test cases is valuable in developing an intuition for the selection of pressure distributions for application of the method in aerodynamic design. For each Mach number pressure drag coefficients are given in table 7 for all the test cases and examples, including that of the 3/4 power law shape. The corresponding d_{max} (maximum body diameter) and fineness ratio (l/d) are also included. Test cases I and III have large drag coefficients because of the body shape obtained. The ogive cylinder of fineness ratio 3 also has a relatively high drag coefficient (twice that of the 3/4 power law shape for the same Mach number and fineness ratio).

The surface pressure-body shape rules and convergence accelerating technique proposed during the development of the axisymmetric inverse design method in this chapter provide the foundation for a general three dimensional inverse method for the supersonic and hypersonic flow regimes. The extension of the method to three dimensions will be described in the next chapter.

V. INVERSE DESIGN METHOD FOR THREE DIMENSIONAL BODIES

The three dimensional inverse method described in this chapter is an extension of the method which was developed for axisymmetric bodies for the Euler equation in Chapter IV. The basic idea is to modify the body geometry along surface streamlines using the same surface pressure-body shape rules. However the extension to three dimensions results in a method that is quite different from that of the axisymmetric case.

The following key aspects of the procedure must be considered in extending the previous method to obtain a general three dimensional inverse procedure:

- A grid point at the k -th iteration is not a grid point at the $k+1$ st iteration. Because both y and z components of each grid point in the cross plane change during the inverse iterations, numerical interpolation and curve fitting are inevitably required. These interpolation procedures must be robust. In addition, the specification of the target pressure distribution requires special consideration.
- The body slope angle and its sign (positive or negative), which can be clearly defined in the two dimensional or axisymmetric case, should be newly defined because the slope of the surface streamline cannot be described by only one body slope angle.

In this chapter, a new inverse method, which accounts fully for three dimensional flowfields and body shapes as described above, will be developed for supersonic and hypersonic bodies. Convergence acceleration techniques will be proposed and demonstrated, and the method will be applied to two three dimensional body test cases with angle of attack.

1. Description of the Method

1.1. General Approach

Two approaches for the three-dimensional inverse method can be considered; one is correcting the body geometry on the axial surface grid line, and the other on the surface streamline. In two-dimensional or axisymmetric flow there is no difference between the axial grid line and the streamline at the surface, but these two are different in three dimensional flow.

In the approach using the grid line, the new surface grids are obtained directly from the inverse procedure. In this case no numerical interpolation and curve fitting is needed. Moreover, the method developed for an Euler analysis computer code can be applied directly to the viscous flow when PNS assumptions are satisfied (there is no difference between the method for the Euler analysis code and for PNS code because surface velocity components are not considered in this approach). However there is no physical basis for the surface geometry correction along a grid line, and thus it is not certain whether this approach works unless numerical experiments are carried out for a specific class of designs of interest.

In the “streamline approach” surface geometry is adjusted along the surface streamline, providing a rational basis for adjustment of the shape to attain a specified pressure distribution (we want to apply the two dimensional pressure-shape rule to the actual three dimensional body modification. Davis³² also used this approach in designing a supersonic conical wing). In this approach, numerical interpolation and curve fitting is needed for each inverse iteration because the coordinates of the modified body are not the grid points. Moreover, some modification to the method is needed to apply this approach to viscous flow because the velocity components at the body surface vanish (the streamline at the outer edge of the boundary layer like portion of the viscous shear layer would have to

be determined). But, as the surface pressure-body geometry rules⁸⁶ are derived from relatively simple two-dimensional shape, this approach is the physically reasonable way of modifying three-dimensional body geometry from a two-dimensional surface pressure-body geometry rule.

The present inverse method will be applied to the Euler equations, and the “streamline approach” is considered and developed because of its physically understandable basis. A feature of the method is that the planform definition y_{max} is obtained during the inverse iteration rather than given as an input (if it is fixed or given as an input, it will not be a three-dimensional problem in a strict sense). The initial body shape can be given arbitrarily, so several different initial body shapes will be considered in the method development stage, even though it is better to assume an initial body shape using surface pressure-grid point relations at the previous planes to accelerate the convergence.

As the planform shape changes in each inverse iteration, the specification of target pressure distribution is not simple. Several ways of specifying the target pressure can be considered;

- using circumferential angle ϕ
- using circumferential distance s
- using y coordinate directly

The first two methods are not appropriate in the present inverse procedure because of the difficulty of getting the x components of the grid points, which are inputs for the interpolation and curve fitting which will be explained later. The target pressure will be given according to the nondimensionalized y coordinates (with respect to the semispan length in each cross flow plane). That means, the target pressure locations change as the body shape changes.

Two convergence criteria are introduced. One is the tolerance for maximum difference

of the pressure coefficient to the target C_p across the crossplane and the other is the average of root mean square of that difference.

$$\Delta C_{p \max} = \text{Maximum of } \left| C_p^T - C_{p_j} \right|, \text{ for } j = 1, \text{jdim} - 1 \quad (5.1)$$

$$\text{and, } \Delta C_{p2} = \frac{\sqrt{\sum \left(C_p^T - C_{p_j} \right)^2}}{\text{jdim} - 1} \quad (5.2)$$

The typical values of these criteria are, $\Delta C_{p \max} < \delta_1 = 0.001$, $\Delta C_{p2} < \delta_2 = 0.0005$. The general procedure of the method is given in flowchart form in figure 34.

1.2. Method in Detail

The details of the three-dimensional inverse procedure are shown in figure 35. Figure 35 corresponds to the box designated “Three Dimensional Inverse Procedure” in figure 34 and this is an iterative procedure requiring the following steps. The surface geometry is adjusted in the plane of the surface streamline (the plane which encloses the points A, B, C at figure 36a and 36b). Point A is cell-centered point in i -1st cross plane, which is known. Point B is a point located in i -th cross plane, which has the same y and z coordinates as point A. Detailed descriptions for the inverse computer code and the input conditions are given in Appendix C.

1.2.1. Definition of θ_0

The slope angle θ of the three dimensional body is usually defined as an angle between surface normal vector and free stream velocity vector.

$$\sin \theta_a = \frac{\vec{V}_\infty}{|\vec{V}_\infty|} \cdot \hat{n} \quad (5.3)$$

However this definition of θ is not adequate for the present inverse procedure because the body shape is modified on the surface streamline and the surface streamline can not be uniquely determined by θ_a alone. Thus, the body slope angle θ should be defined in a different way.

- θ is defined as an angle between the streamline and the x -axis.

This angle is used for body modification and θ_a is used to calculate $A(M_L, \theta_a)$, which is the relation between changes in pressure and changes in slope previously investigated in Chapter IV. Since $A(M_L, \theta_a)$ is derived from local inclination method, $A(M_L, \theta_a)$ is defined by the equation

$$\Delta C_p = A(M_L, \theta_a) \Delta \theta \quad (5.4)$$

For the zero angle of attack case, there is typically only a small difference between θ and θ_a . The converged body geometry can be obtained using only θ . For the nonzero angle of attack case, the effect of θ_a is so large that a converged body geometry will not be obtained without considering θ_a . The body slope angle θ is expressed by surface velocity vector:

$$\theta_{\theta_j} = \cos^{-1} \left[\frac{u_j}{\left(u_j^2 + v_j^2 + w_j^2 \right)^{\frac{1}{2}}} \right] \quad (5.5)$$

Next consider y_C, z_C (y, z coordinates of the point C in figures 36a and 36b). AC is the surface streamline which passes through the given point A. C is the intersection point of

the surface streamline to the vertical plane of the i -th station. The values of y_C , z_C can be easily determined using surface velocity components u_j , v_j , w_j .

$$y_{C_j} = y_B + \frac{\Delta x}{2} \cdot \frac{v_j}{u_j} \quad (5.6)$$

$$z_{C_j} = z_B + \frac{\Delta x}{2} \cdot \frac{w_j}{u_j} \quad (5.7)$$

1.2.2. Sign of θ_0

As can be seen in equation (5.5), the sign of θ_0 is not determined. The sign will be found by considering the length from the point O' as shown in figure 36b.

$$\text{i.e. } \theta_{0_j} = \theta_{0_j} \text{ for } r_C \geq r_B \quad (5.8)$$

$$\theta_{0_j} = -\theta_{0_j} \text{ for } r_C \leq r_B \quad (5.9)$$

This procedure is needed because we want to apply local two dimensional theory along a streamline in the actual three-dimensional case.

1.2.3. $A(M_L, \theta_a)$ and θ_1

θ_1 is obtained by the surface pressure-body geometry rule, which was thoroughly investigated in the last chapter. Two surface pressure-body geometry rules which have good convergence behavior⁸⁶ are selected; i.e. shock-expansion rule and tangent-cone rule.

- Shock-expansion pressure-shape rule

$$A(M_L, \theta_a) = m \cos \theta \cdot \left(\frac{2}{M_L} + (\gamma + 1)m \sin \theta_a + \frac{3}{16}(\gamma + 1)^2 M_L m^2 \sin^2 \theta_a \right) \quad (5.10)$$

- Tangent cone pressure-shape rule

$$A(M_L, \theta_a) = \tau \left(2\theta_a + \frac{1}{2}c\theta_a^{-\frac{1}{2}} \right) \quad (5.11)$$

m , τ , and c are given in equations (4.19), (4.8), and (4.21).

In each body point, θ_1 can be calculated from,

$$\theta_{1j} = \theta_{0j} + \frac{C_p^T - C_{p_j}}{A(M_L, \theta_a)} \quad (5.12)$$

1.2.4. Corrected Body Geometry (y_D, z_D)

y_D, z_D (y, z coordinates of the point D in figures 36a and 36b) are the corrected surface geometry by the new surface slope angle θ_1 . In figures 36a and 36b, the points B, C and D lie on the same line and line AB is perpendicular to line BD so, the coordinate y_D, z_D are easily obtained by,

$$y_{Dj} = y_{Bj} + dy_{Dj} \quad (5.13)$$

$$z_{Dj} = z_{Bj} + dz_{Dj} \quad (5.14)$$

$$dy_{Dj} = \pm \frac{\tan \theta_{1j}}{\sqrt{(1 + P_j^2)}} \cdot \frac{\Delta x}{2} \quad (5.15)$$

$$dz_{D_j} = P_j \cdot dy_{D_j} \quad (5.16)$$

$$P_j = \frac{z_{C_j} - z_{B_j}}{y_{C_j} - y_{B_j}} \quad (5.17)$$

To select only one value from equation (5.16) and (5.17), the distances from x axis are calculated for points B, C, D_P , D_M . These distances are denoted by r_B , r_C , r_{D_P} , r_{D_M} , respectively. By comparing r_{D_P} , r_{D_M} with r_B and by considering the sign of θ_1 , D_P or D_M is selected as a new body point.

1.3. Methods of Accelerating the Convergence

With the procedure described above, the converged body shape can be obtained. However, 10 inverse iterations are required for most sample test cases, as explained in the next section. Thus, a dramatic improvement in the convergence behavior is required. Rapid convergence is an important consideration in developing a practical method. Four ways are considered as follows,

1.3.1. Reduced grid density

Even though the number of grid points at each crossplane should be large enough to obtain an accurate body shape with the worst distribution of the interpolation input points, consideration was given to reducing the grid density because dense three dimensional grid calculations usually require large CPU times. Crossplane grids of 41×30 and 31×20 are examined in the next section.

1.3.2. Intermediate design solution not fully converged

The usual convergence criterion adopted for the analysis case is a 4 order of magnitude decrease in the residual. This takes a large CPU time, especially to go from a 10^{-3} to a 10^{-4} decrease of the residual. Thus it is proposed to use a convergence criterion which is not strict (two and one half order of magnitude decrease in residual). In other words, solutions (velocities, pressures and local mach numbers) which were not fully converged are used to begin the next inverse iteration. As the inverse iterations progress, the starting values of the residual decrease. Thus the present convergence criterion provides quite accurate results in the final inverse iteration and in addition reduces the CPU time.

1.3.3. Regula-falsi root finding scheme

By applying the regula-falsi root finding scheme to the axisymmetric inverse problems in the previous chapter, the convergence behavior was dramatically improved. Therefore, the regula-falsi scheme is introduced again and demonstrated for the three dimensional example calculations.

1.3.4. Iteration in real sense

As the surface pressure-body geometry rule is basically derived from the axisymmetric bodies of revolution, some modification in the method is needed to accelerate the convergence when the effect of angle of attack is considered. In the normal operation of the method, θ_0 at $k+1$ st iteration is obtained from the updated body geometry, not from θ_1 at k th iteration. The method to use $\theta_1^{(k)}$ to $\theta_0^{(k+1)}$ is introduced when ΔC_{p2} is relatively small (typically $\Delta C_{p2} < 0.005$).

With these four convergence acceleration techniques the convergence is attained

10~20 times faster. This results in a tremendous savings in CPU time. All these methods are applied to the test cases and will be discussed in detail in the next section.

2. Example Calculations and Results

The new three dimensional inverse procedure which was developed and explained in the previous section is now applied to test cases in the supersonic ($M_\infty = 3.0$) and hypersonic ($M_\infty = 6.28$) flow regime as did in the previous chapters.

Two body geometries (elliptic cone and 'wing like' nonconical cambered body) are considered with and without angle of attack. 41x30 and 31x20 crossplane grids were selected and run based on the previous work in Chapter II (grid density convergence was studied for the power law bodies over a wide Mach number range). VPI & SU IBM 3090 supercomputer was used again. Typical CPU times for 1 cross flow plane calculations of both test cases were 5~6 minutes for 41x30 grid and 2~3 minutes for 31x20 grid system.

Using the relatively simple body geometry (elliptic cone) to establish a target pressure distribution, several numerical experiments are performed. Using the information from these numerical experiments, the method has been refined.

- The convergence acceleration techniques which were proposed in the previous section (reducing the grid density, using not fully converged solution, applying the regula-falsi root finding scheme and iteration in real sense) are applied and tested one by one.
- By considering several different initial body geometries for the same target pressure distribution, the capability of the method to find the planform shape and the exact body geometry is demonstrated.
- The effect of different surface pressure-body geometry rules is investigated.
- The capability of the method to find the exact body shape in the presence of the angle of attack ($\alpha = 10^\circ$) is demonstrated.

From the example calculation of the ‘wing like’ nonconical cambered body, the capability of the method to find the body geometry which has an abrupt pressure change across the cross plane, especially in the case of angle of attack, is investigated and hence, it will be demonstrated that present method is in fact robust and useful for the aerodynamic design of three dimensional bodies in the supersonic to moderate hypersonic flow regime.

Before making inverse calculations, our application of the basic analysis code was validated by making an analysis computation and comparing the results with previously obtained results. The calculation was performed for an elliptic cone (ratio of major axis length to minor axis length, $a/b = 2.0$) at $M_\infty = 5.8$ and $\alpha = 10^\circ$. As shown in figure 37, the pressure coefficients at the body surface are in good agreements with the experimental results⁸⁷ except in the region of separation ($15^\circ < \varphi < 50^\circ$). These results were previously used by Siclari and Rubel⁸⁸ to validate a computational technique for correcting potential solutions. Having verified the basic accuracy of this analysis code, the inverse calculations for two test cases are undertaken.

2.1. Elliptic Cone Testcase ($a/b= 2.0$)

The elliptic cone is the simplest body shape which is not axisymmetric, and thus is a good candidate for the method evaluation and development phase of the work. The convergence acceleration techniques discussed above were tested, and the results are shown at figures 38 and 39. For two different initial body geometries ($y_{max} = 0.26663$ for figure 38, $y_{max} = 0.4$ for figure 39) at $M_\infty = 6.28$ and $\alpha = 0^\circ$, the effects of grid density and regula-falsi scheme are investigated. Only a small improvement is obtained using the higher grid density. The advantages of the regula-falsi scheme are clearly shown. As two normal iterations of the inverse method are needed to apply the regula-falsi scheme, ΔC_p^2 at the first two iterations with and without regula-falsi scheme are the same, and ΔC_p^2

decreases rapidly as the number of iterations increases. It takes only 4 ~ 5 inverse iterations to obtain a converged body geometry using regula-falsi. Even with the normal operation of the method, the converged body geometry is obtained within 10 inverse iterations (When the convergence accelerating techniques explained in section 1.3. were applied, CPU time is 1.6~6.5 minutes. Otherwise, it goes to 27~43 minutes. More than 10 times saving of CPU time can be obtained by using these techniques!) The convergence of the body geometry in each inverse iteration is shown at figure 40 for initial $y_{max} = 0.2$. During the iteration, body geometry changes arbitrarily and goes to the exact body geometry within 6 inverse iterations. From figure 38, 39, 40, it is shown that initial body geometry can be given arbitrarily and the convergence accelerating techniques proposed before are working very well.

The effect of surface pressure-body geometry rules is investigated for the same body and Mach number and shown in figure 41. The tangent-cone rule has slightly better convergence behavior than the shock-expansion rule but basically the trend is the same. As described in the previous chapter, the tangent-cone rule has problems for negative body slope regions. Therefore, the shock-expansion rule is adopted in spite of the slightly better convergence behavior of the tangent-cone rule.

The effect of angle of attack is considered for $M_\infty = 6.28$. Body geometry and C_p convergence behavior are shown in figures 42 and 43. The convergence acceleration techniques (including the iteration in real sense) are also applied for this inverse calculation. The results show that even with the presence of peak pressure, the converged body geometry is found within 4 iterations at the lower surface. It takes a little longer (about 6 iterations) on the upper surface. The main reason for this behavior arises from the surface pressure-body geometry rule, which was derived at zero angle of attack. Convergence behaviors with and without regula-falsi scheme is shown in figure 44.

For several initial body geometries ($y_{max} = 0.2, 0.26663, 0.4$), the present inverse method is applied at supersonic speed ($M_\infty = 3.0$) with and without regula-falsi scheme.

The results are shown in figures 45 and 46 for the case with initial $y_{max} = 0.4$. Converged body geometries were obtained within 6 ~ 8 iterations without regula-falsi scheme and 4 ~ 6 iterations using the regula-falsi scheme for these three different initial body geometries.

2.2. 'Wing Like' Nonconical Cambered Body

The nonconical cambered body is a relatively simple three dimensional body, but it has a rapid decrease of surface pressure near the attachment line as the flow expands around the edge. Thus, this body geometry is a good test case for the three dimensional inverse procedure development and provides a demonstration of its robustness. Conically cambered shapes at supersonic speed were investigated in detail in reference 89.

To reduce the computation time, 3/4 of the body length from the nose has an elliptic cone shape and the remaining portion of the body is a 3/4 power law shape which is not conical. The cross sectional body geometry in each marching plane is determined by,

- camber angle at y_{max} point
- base elliptic cone ($a/b = 3.0$)
- 3/4 power law shape along the x -axis

The camber angle is varying from 0° (elliptic cone) at $x/l = 0.75$ to -12° at $x/l = 1.0$ with a cosine curve. The computational grids and the cross-sectional body shape at $x/l = 1.0$ (camber angle = -12°) is shown in figure 47. 41 crossplane grid points are used to cover the sharp decrease of the pressure near y_{max} . 6 stations are constructed for the nonconical axial portion of the body and the present inverse procedure is applied only to the last station. Details for the grids used are described in Appendix B.

All the convergence accelerating techniques are applied to this test case. The body geometry convergence at zero angle of attack is shown at figure 48. The converged body

shape is obtained within 8 iterations and correctly reproduces the target pressure distribution. There is no big difference in convergence behavior between the case with regula-falsi scheme and the case without it (slightly better in the case with regula-falsi but no difference in the number of iterations with the convergence criteria used).

The effect of angle of attack is shown at figures 50, 51, and 52. The number of iterations to get a converged body geometry is slightly higher than that of the zero angle of attack case (9 inverse iterations). As shown at figure 51a and 51b, the sharp decrease in pressure near y_{max} is correctly reproduced as the iteration proceeds. Moreover, this result shows that the method has no difficulties in finding the body shape on the upper surface, which has negative pressure coefficients over the entire region. The convergence behavior including angle of attack has nearly the same trend as that of zero angle of attack.

Even though the CPU time depends on the initial guess of the body geometry, free stream Mach number, and the inverse calculation options(regula-falsi etc.), typical CPU times are 2~7 minutes for elliptic cone test case and 5~10 minutes for 'wing like' nonconical body test case.

In this chapter, the method to modify the body geometry along the surface streamline was developed and techniques to accelerate the convergence were proposed with the new definition of body slope angle. In order to verify the usefulness of the developed three dimensional inverse method in the practical aerodynamic design problems, the inverse solution uniqueness and the existence will be discussed in the next chapter.

VI. INVERSE SOLUTION UNIQUENESS AND THE DOMAIN OF ITS EXISTENCE

In Chapter IV and V, inverse procedures for axisymmetric and three dimensional supersonic and hypersonic bodies have been developed and shown to be very robust and efficient. The next question concerns the issue of whether a unique solution (body geometry) exists for the given target pressure distribution.

As mentioned in Chapter I (Introduction), current two dimensional inverse Euler methods and general three dimensional inverse procedures do not have any mathematical or numerical criteria for the existence and uniqueness of the solution. However, useful predictions for the existence of the inverse solution and the verification of its uniqueness can be made for the three dimensional inverse procedure⁹⁰ developed for the Euler computer code and will be described in this chapter.

1. Domain of Inverse Solution Existence

As the three dimensional inverse method⁹⁰ uses space marching technique for the supersonic and hypersonic free stream speeds, it steps one plane at a time, using the same approach at each station. As pointed out in reference 86, we are not concerned with the existence of the resulting body a priori because the body shape is found in each marching plane, and the solution can simply be halted automatically if the specified pressure distribution leads to a body that would require a negative radius according to the calculation. Thus the target pressure distribution at a current computation station may be changed during the inverse iterations in case a negative body thickness is predicted.

Even though general mathematical formulation or numerical criteria for the existence of inverse solution in two dimensional Euler or three dimensional problems are not possible at present, the domain of inverse solution existence can be predicted at least for the supersonic and hypersonic bodies which employ space marching techniques with and without an angle of attack using simple surface pressure - body shape rules.

The idea is to consider each grid point in the current cross plane as a corresponding axisymmetric point which has the same z_i (z -component at i -th cross plane) and simply calculate the body slope angle which makes the thickness zero. At every grid point in the current cross plane, the body slope angle can be easily calculated by,

$$\tan \theta_j = \left(\frac{dz}{dx} \right)_j \approx \frac{(z_{i+1} - z_i)_j}{\Delta x_i} \geq \frac{(z_0 - z_i)_j}{\Delta x_i} \quad \text{for } j = 1, j_{\text{dim}} \quad (6.1)$$

where, z_0 is the reference point value (z_0 is set to zero for the elliptic cone test case and is set to z value at y_{max} for the 'wing like' nonconical cambered body test case considered below). From the relation below for each grid point in the i -th cross plane, it can be said that the pressure distribution which is greater than the pressure distribution at θ_{extreme} must be prescribed for the inverse solution to exist for all the grid points in the current (i -th) computational cross plane.

$$C_p(\theta) \geq C_p(\theta_{\text{extreme}}) \quad \text{where, } \theta_{\text{extreme}} = \tan^{-1} \left(\frac{z_0 - z_i}{\Delta x_i} \right) \quad (6.2)$$

This approach is applied to the elliptic cone and 'wing like' nonconical cambered body which were tested in the last chapter. For several elliptic body shapes, the C_p distributions are calculated for $M_\infty=6.28$ and zero angle of attack. As shown at figure 53, the predicted limit of C_p is less than the C_p 's of all the body shapes which were considered including the body shape of zero thickness at the i -th plane. From this, it can be said that no target pressure distribution which has smaller value than the predicted limit will generate a body shape. The same approach is also applied for the angle of attack 10° . As shown in

figure 54a for upper surface and 54b for lower surface, the same prediction for the target pressure distribution can be made in the presence of angle of attack.

For ‘wing like’ nonconical cambered body shape, C_p distribution is calculated for the body shape of zero thickness at the i -th plane using the **cfl3de** analysis code. The predicted results and the calculated extreme C_p distribution are compared in figure 55a for upper surface and 55b for lower surface. Previous calculated result for the body shape considered in Chapter V is also included for reference. The same conclusion as those of the elliptic cone test case can be made for this cambered body.

Even though this approach to predict the domain of inverse solution existence is limited to the space marching problem at supersonic and hypersonic speeds, it is very simple to apply and, without any inverse calculation, provides the necessary condition for the inverse solution existence, which will help to specify the attainable target pressure distribution for the supersonic and hypersonic body design.

2. Inverse Solution Uniqueness

In the three dimensional inverse procedure⁹⁰, the pressure distribution which is non-dimensionalized with respect to the semispan length in the cross plane is used as a target pressure distribution in order to find the body geometry which includes determination of the planform shape. The planform shape is to be found as part of the solution during inverse iterations. In the three dimensional inverse examples considered above, a unique body geometry was found for several different guesses of the initial body geometry. This leads to the following question: what happens if the target pressure distribution is the same as one of the three dimensional inverse examples but the planform shape is fixed (i.e. semispan length is given as a design input)? Can a converged solution be obtained to satisfy the target pressure distribution while keeping the given semispan length? Can a converged body geometry be obtained by correcting just the contour distribution i.e. by

changing the cross sectional shape while maintaining the semispan length in the cross plane. These questions are basically a problem of inverse solution uniqueness. The question is summarized as; Is the body shape unique which satisfies both the non-dimensionalized target pressure and the previous plane data?

In inviscid flow, there may exist numerous solutions which satisfy the non-dimensionalized target pressure distribution by similarity (similar body shapes will have the same non-dimensionalized pressure distribution at the same free stream condition). Thus, an infinite number of body shapes can be generated for the given non-dimensionalized target pressure distribution. Even though the cross-sectional shape of a three dimensional body is similar in concept to the shape in two dimensions, especially when the space marching numerical technique is employed, there is an influence from the previous data plane on the present plane. That means, the inverse solution (section geometry) in the current plane is dependent upon not only the non-dimensionalized pressure distribution but also solutions at the previous plane (this is a general characteristic in three dimensional flow. In analysis solutions the flow properties in the present cross plane are dependent upon both the solution at the previous plane and the body geometry data in the current plane).

In two dimensional or axisymmetric flow, a new grid point z_i (z-component at i -th cross plane) is determined by the surface slope angle at the current cross plane and z_{i-1} (z-component at i -1st plane). z_{i-1} is known from the geometry data at the previous plane and the surface slope angle is determined by the specified pressure using a surface pressure-body geometry rule. If C_p is a single-valued function of surface slope, only one θ will be found from the specified pressure coefficient. Then the body geometry point z_i is uniquely determined. If the surface pressure-body geometry rule is exact, a unique body shape will be found without any inverse iterations. Thus, first consider the relation between the surface pressure and body slope angle θ to show the uniqueness of the solution. The first step is to show that $C_p(\theta)$ is a single-valued function of θ . The surface pressure -

body geometry rules which were examined before in the previous chapters are considered again with their exact forms.

- Shock-expansion pressure-shape rule

$$C_p = \frac{\gamma+1}{2} m^2 \sin^2 \theta + \frac{2}{M_\infty} m \sin \theta \sqrt{1 + \left(\frac{\gamma+1}{4}\right)^2 M_\infty^2 m^2 \sin^2 \theta} \quad (6.3)$$

- Tangent cone pressure-shape rule

$$C_p M_\infty^2 = \frac{4}{\gamma+1} (k_0^2 - 1) + (k_0 - k_1)^2 \frac{2(\gamma+1)k_0^2}{2 + (\gamma-1)k_0^2} \quad (6.4)$$

$$\text{where , } k_1 = M_\infty \theta \quad (6.5)$$

$$k_0 = \frac{\gamma+1}{\gamma+3} k_1 + \sqrt{\left(\frac{\gamma+1}{\gamma+3}\right)^2 k_1^2 + \frac{2}{\gamma+3}} \quad (6.6)$$

C_p is shown as a function of the body surface slope in figure 56a and 56b. As can be seen in the figure, only one value of θ is matched for the given surface pressure coefficient for both supersonic and hypersonic speeds. As described in the previous chapter, the basic idea of the three dimensional inverse procedure is to apply the surface pressure - body geometry rules along the surface streamline. The surface geometry is then adjusted in the plane of the surface streamline (the plane which encloses the points A, B, C at figure 36a and 36b). From the $C_p - \theta$ relation, there is only one body surface slope θ_j for the given C_{p_j} , so, the new body geometry point at the present plane (i -th plane) is uniquely determined by,

- body surface slope θ_j
- the geometry data at i -1st data plane

In the figure 36a and 36b, AC is the surface streamline which passes through the given point A. C is the intersection point of the surface streamline to the vertical plane of the i -th station (points B, C and θ_0 were determined by solutions of the previous plane and only one value of θ_1 is obtained by the pressure - body shape rule). Using this information, the new surface point D is uniquely determined (as explained in the previous chapter, by comparing r_{D_P} , r_{D_M} with r_B and by considering the sign of θ_{1_j} , either D_P or D_M is selected as a new body point).

These new geometry points at i -th plane are the exact ones if

- the surface pressure-body geometry rule is exact for the corresponding body shape and for given free stream conditions.
- there is no three dimensional effects (the neighboring points at the same plane affect the solution at the point considered)
- there is no error during the numerical interpolation and curve fitting

As the inverse iteration continues, ΔC_p (the difference between the target pressure coefficient and the calculated pressure coefficient) is decreasing and the body shape goes to the exact one. The planform shape modification (point F at figure 36a) is exactly the same procedure with other points like C and D.

Because the inverse solution which satisfies both target pressure distribution and the body geometry data at the previous cross plane is unique as explained above, the same inverse solution must be obtained regardless of the initial guesses for the body geometry (In the last chapter, many inverse calculations were made for the elliptic cone with different initial guesses); As the desired body geometry (i.e. inverse solution) is unique, the corresponding y_{max} point is also unique. Thus, if the initially assumed y_{max} point happens to be the same with the desired (or, exact) y_{max} point, the point should not move during the inverse iteration for the solution to be unique. Even though the developed inverse method⁹⁰ allows for the body points to move to any direction, the body geometry must be modified

only along the vertical direction because the planform shape (y_{max} point) will not be changed during the inverse iterations. To check whether the present three dimensional inverse procedure has the aforementioned phenomenon, which implies the uniqueness of the solution, a sample inverse calculation is performed for the elliptic cone test case which was investigated in the previous chapter with the initial guess of exact planform shape (it is different only in thickness distribution along the cross plane as is shown in figure 57). Even though no restriction to the direction of body shape correction is given, the behavior is similar to the case with fixed planform and the converged body geometry is found within 7 inverse iterations. The body geometry convergence is shown in figure 57.

In general, the planform shape (y_{max}) which satisfies a given target pressure distribution is not known until an inverse calculation is performed. Thus, in case the planform shape is prescribed as a design requirement, an understanding for the effect of the target pressure selection on the resulting planform shape is needed. Then the target pressure distribution may be modified during the inverse iterations to satisfy the given planform shape. The effect of the target pressure selection on the resulting planform shape is investigated for the 'wing like' nonconical cambered body which was calculated in the previous chapter. Body shapes which are different mainly in semispan length are calculated at $M_\infty=6.28$ (figure 58). The corresponding differences of the pressure distributions are clearly shown in figure 59a and 59b with and without the angle of attack. Even though these body shapes are three dimensional, the local changes of the body shapes result in local changes of corresponding pressure distributions as can be seen in the figures. Thus, by adjusting the target pressure distribution near $y/a=1.0$ (a is semispan length), a body shape which has a different semispan length can be generated. This kind of investigation is useful for practical aerodynamic design problems (By modifying the specified target pressure distribution non-dimensionalized with respect to the semispan length during the inverse iterations, the developed three dimensional inverse method can be applicable to the design problem of fixed planform shape without any restriction to the inverse code).

One other example calculation is performed for the same body considered above at the

angle of attack 10° to support the uniqueness of the inverse solution . Two different data in the previous plane (solutions for zero angle of attack and those at 10°) are used for the inverse routine with the same target pressure distribution. The result is shown in figure 60. From this figure, it is shown that different data in the previous plane result in different body shapes even though the target pressure distributions are the same.

Even though these examples have been tested only for the space marching problem, these are the examples to support the general explanation described above for the uniqueness of the inverse solution.

In order to verify the usefulness of the three dimensional inverse method developed in Chapter V, inverse solution existence and uniqueness were discussed with several example calculations. An investigation of the effect of planform shape change on the pressure distribution was made, which is useful in practical aerodynamic design problems.

VII. CONCLUDING REMARKS

New inverse methods for axisymmetric and three dimensional body design have been developed in the supersonic and hypersonic flow regimes where a space marching numerical technique is valid. The first objective of the inverse method is to find a body shape which has aerodynamically desirable features of the flow field such as minimization of the wave drag, and delay of boundary layer separation and laminar to turbulent transition. This objective can be achieved by proper specification of the surface pressure distribution. In order to get a general understanding for the specification of target pressure distributions required for the development of the inverse procedures, minimum drag body shapes have been investigated in detail in the supersonic and hypersonic flow regimes with advanced computer code, **cfl3de**.

After reviewing previous studies for the minimum drag bodies in the supersonic and hypersonic flow regimes, numerical investigations, especially for the power law shapes with specified fineness ration (l/d), have been made. The effect of grid density on the numerical accuracy was studied first and then the actual numerical analysis for these bodies was made. Numerous power law shapes were numerically evaluated in the free stream Mach number ranged from 3 to 12. The newly obtained results for the supersonic and hypersonic minimum drag body study are summarized as follows,

- The optimum power law is $n = .69$, virtually independent of Mach number for the $l/d = 3$ case considered. For $l/d = 5$ the minimum changes slightly, occurring at $n = .70$.
- Use of either the calorically and thermally perfect gas model or the equilibrium air model produces the same results for the low drag bodies studied over a Mach number range up to $M_\infty=12$. This result, while not surprising to workers in the field, has apparently not been well documented.

- The body shape proposed by Hayes has been shown to have a higher drag than the optimum power law body.
- Effects of viscosity are negligible in the determination of minimum drag shapes for which the flow disturbance has been minimized.
- The blunt noses on minimum drag bodies are confined to such a small region that the detailed treatment of the nose is not necessary to obtain valid results for the flow over the rest of the body.

Using simple Newtonian flow theory and a detailed numerical investigation of the blunt noses on minimum drag bodies, a new criterion for the definition of an aerodynamically sharp shape is proposed. The relevant new findings are summarized as follows,

- Aerodynamically sharp shapes are different than geometrically sharp shapes. The condition for aerodynamic sharpness is that the leading edge should behave as x^n where $n > 1/2$!
- The investigation shows that an aerodynamically sharp shape is a shape that is $O(x^n)$, where $n > 1/2$. Thus, the minimum drag power law shapes have low drag and behave as though they are sharp even though they are not geometrically sharp. The Karman Ogive and Sears-Haack bodies are also aerodynamically sharp using this criteria.
- In considering classical hypersonic and supersonic shapes it is found that the radius and curvature of the supersonic shapes (Karman Ogive and Sears-Haack body) are $O(x^{3/4})$ near the nose. Hence they are directly related to the hypersonic result. Linearized supersonic and nonlinear hypersonic analysis both lead to the same result!
- The issue of “how sharp is sharp” when considering the specification of contours for model fabrication and manufacturing has been resolved. Aerodynamically sharp shapes do not have to be geometrically sharp.

With the basic knowledge for the surface pressure distributions over supersonic and

hypersonic minimum drag bodies, an inverse design method for axisymmetric bodies has been developed. Several surface pressure-body geometry rules have been examined and the most robust and fastest rule suitable for inverse calculations has been identified: The shock expansion pressure-shape rule can be applied to regions of negative body slope and negative pressure coefficient distribution, and hence has wide application. The modified tangent cone pressure-shape rule with regula-falsi root finding scheme shows the fastest convergence for many example calculations. But because of its wide application, the shock expansion pressure -shape rule has been applied for the three dimensional inverse method development. To solve the slow convergence problem in the *residual-correction type* approach a new method which considers the inverse iteration as a root finding problem, was proposed and run successfully for many example calculations. Numerous examples in both supersonic and hypersonic flow were presented: The supersonic and hypersonic minimum drag bodies which were investigated in Chapter II, was used as a test case during the procedure development. Several target pressure distributions were proposed and the corresponding body shapes were successfully found in both supersonic and hypersonic flow regimes. Hence the robustness and efficiency of the procedure was demonstrated.

By applying the surface modification procedure developed for the axisymmetric flow to the three dimensional body along the surface streamline, the inverse method for three dimensional body design in supersonic and hypersonic flow regimes has been developed. The body slope angle for the three dimensional body was newly defined and its sign was determined according to the direction of the surface streamline in each grid point. With the new concept for the body slope angle, the detailed procedure to modify the body geometry along the surface streamline was described and applied to several example calculations. Because three dimensional calculations usually require high computation times, especially in the design mode, special emphasis was given to convergence acceleration techniques in order to make this method a practical design tool: reduced grid density, intermediate design solution not fully converged, regula-falsi root finding scheme used in the axisymmetric inverse method development, and inverse iteration in real sense were proposed and run successfully for the three dimensional example calculations. Tremendous savings in CPU

time resulted. Elliptic cone and ‘wing like’ nonconical cambered body with and without angle of attack have been considered as three dimensional test cases. From these example calculations, the robustness and efficiency of the three dimensional inverse method in both supersonic and hypersonic flow regimes have been demonstrated.

For the three dimensional inverse procedure using a space marching technique, the inverse solution existence and uniqueness problems were discussed. A simple method to predict the domain of the inverse solution existence has been proposed and applied to several example test cases. It gives useful information for the inverse solution existence without any inverse calculation. By investigating the relations between the surface pressure and the body geometry, a general explanation for the inverse solution uniqueness has been described. Sample calculations to support this explanation has been successfully made.

The axisymmetric and three dimensional inverse methods developed during the study are for the Euler analysis computer code in the supersonic and hypersonic flow regimes. Thus, recommended future research work will be directed to the following areas,

- Application of the current inverse methods to viscous flow (PNS) at the supersonic and hypersonic flow regimes.
- Modification of the current inverse methods for application to transonic and subsonic airfoils and wing designs using full potential or Euler analysis codes.
- Coupling of the developed inverse methods with a numerical optimization algorithm, in order to design a desirable body shape directly and efficiently.

In the first two research areas, new surface pressure-body shape rules must be considered, since the current surface pressure-shape rules are for supersonic and hypersonic inviscid flows. In the final research area, specific attention should be paid to providing the shape functions. This can be performed using inverse methods⁶ to maintain efficiency.

REFERENCES

1. Harris, R.V., Jr., "On the Threshold - The Outlook for Supersonic and Hypersonic Aircraft," 52nd AIAA Wright Brothers Lecture, AIAA Paper 89-2071, 1989.
2. Mason, W.H., and Lee, Jaewoo, "On Optimal Supersonic/Hypersonic Bodies," AIAA Paper 90-3072, August 1990.
3. Miele, A.Ed., *Theory of Optimum Aerodynamic Shapes*, Academic Press, New York, 1965.
4. Mason, W.H., and Lee, Jaewoo, "Aerodynamically Blunt and Sharp Bodies," Under Preparation for the 30th Aerospace Science Meeting, January 1992.
5. Rubbert, P., and Goldhammer, M., "CFD in Design: An Airframe Perspective," AIAA Paper 89-0092, January 1989.
6. Aidala, P.V., Davis, W.H., Jr., and Mason, W.H., "Smart Aerodynamic Optimization," AIAA Paper 83-1863, July 1983.
7. Slooff, J.W., "A Survey of Computational Methods for Subsonic and Transonic Aerodynamic Design," Proceedings of the First Int. Conf. on Inverse Design Concepts in Eng. Sci., pp. 1~67, October 1984.
8. Hicks, R.M., Murman, E.M., and Vanderplaats, G.N., "An Assessment of Airfoil Design by Numerical Optimization," NASA TM X-3092, 1974.
9. Hicks, R.M., and Vanderplaats, G.N., Murman, E.M., and King, Rosa T., "Airfoil Section Drag Reduction at Transonic Speeds by Numerical Optimization," SAE Paper 760477, 1976.
10. Hicks, R.M., and Vanderplaats, G.N., "Application of Numerical Optimization to the Design of Supercritical Airfoils without Drag-Creep," SAE 770440, 1977.
11. Hicks, R.M., and Henne, P.A., "Wing Design by Numerical Optimization," AIAA Paper 77-1247, 1977.

12. Haney, H.P., Johnson, R.R., and Hicks, R.M., "Computational Optimization and Wind Tunnel Test of Transonic Wing Designs," AIAA Paper 79-0080, 1979.
13. Vanderplaats, G.N., and Hicks, R.M., "Numerical Airfoil Optimization Using a Reduced Number of Design Coordinates," NASA TM X-73151, July 1976.
14. Vanderplaats, G.N., "An Efficient Algorithm for Numerical Airfoil Optimization," AIAA Paper 79-0079, 1979.
15. Vanderplaats, G.N., "CONMIN - A FORTRAN Program for Constrained Function Minimization," NASA TM X-62, 282, 1973.
16. Bock, K.-W., "Aerodynamic Design by Optimization," AGARD CP 463, No. 20, May 1989.
17. Schone, J., "Design of Supersonic Wings Using an Optimization Strategy Coupled with a Solution Scheme for the Euler Equations," AIAA Paper 90-3060, 1990.
18. Dulikravich, G.S., Buss, R.N., Strang, E.J., and Lee, S., "Aerodynamic Shape Optimization of Hypersonic Missiles," AIAA Paper 90-3073, 1990.
19. Bowcutt, K.G., Anderson, Jr., J.D., and Capriotti, D., "Viscous Optimized Hypersonic Waveriders," AIAA Paper 87-0272, January 1987.
20. Corda, S., and Anderson, Jr., J.D., "Viscous Optimized Hypersonic Waveriders Designed from Axisymmetric Flow Fields," AIAA Paper 88-0369, January 1988.
21. Lin, W.F., Chen, A.W., and Tinoco, E.N., "Three Dimensional Transonic Nacelle and Winglet Design," AIAA Paper 90-3064, 1990.
22. Stratford, B.S., "The Prediction of Separation of the Turbulent Boundary Layers," *J. Fluid Mech.*, 5, pp. 1~16, 1959.
23. Stratford, B.S., "An Experimental Flow with Zero Skin Friction throughout Its Region of Pressure Rise," *J. Fluid Mech.*, 5, pp. 17~35, 1959.
24. Fray, J.M.J., Slooff, J.W., Boerstoeel, J.W., and Kassies, A., "Inverse Method with Geometric Constraints for Transonic Aerofoil Design," *International Journal for Numerical Methods in Engineering*, Vol. 22, pp. 327~339, 1986.

25. Zedan, M.F., and Dalton, C., "Potential Flow around Axisymmetric Bodies: Direct and Inverse Problems," *AIAA Journal*, Vol. 16, No.3, pp. 242~250, March 1978.
26. Johnson, F.T., and Rubbert, P.E., "Advanced Panel-Type Influence Coefficient Methods Applied to Subsonic Flows," AIAA Paper 75-50, January 1975.
27. Tranen, T.L., "A Rapid Computer Aided Transonic Airfoil Design Method," AIAA Paper 74-501, 1974.
28. Volpe, G., "Two-Element Airfoil Systems Design: An Inverse Method," AIAA Paper 78-1226, 1978.
29. Arlinger, B., and Schmidt, W., "Design and Analysis of Slat Systems in Transonic Flow," ICAS Paper, 1978.
30. Carlson, L.A., "Transonic Airfoil Analysis and Design Using Cartesian Coordinates," *J. Aircraft*, Vol. 13, pp. 349~356, May 1976.
31. Henne, P.A., "An Inverse Transonic Wing Design Method," AIAA Paper 80-0330, 1980.
32. Davis, W.H. Jr., "Technique for Developing Design Tools from the Analysis Methods of Computational Aerodynamics," *AIAA Journal*, Vol.18, No.9, pp. 1080~1087, September 1980.
33. Garabedian, P., and MaFadden, G.B., "Design of Supercritical Swept Wings," *AIAA Journal*. Vol. 20, No. 3, pp. 289~291, March 1982.
34. Malone, J.B., "A Subsonic Panel Method for Iterative Design of Complex Aircraft Configuration," *J. Aircraft*, Vol. 19, pp. 820~825, October 1982.
35. Malone, J.B., Vadyak, J., and Sankar, L.N., "Inverse Aerodynamic Design Method for Aircraft Components," *J. Aircraft*, Vol. 24, pp. 8~9, January 1987.
36. Takanashi, S., "An Iterative Procedure for Three Dimensional Transonic Wing Design by the Integral Equation Method," AIAA Paper 84-2155, August 1984.

37. Tatsumi, S., and Takanashi, S., "Experimental Verification of Three Dimensional Transonic Inverse Method," AIAA Paper 85-4077, October 1985.
38. Ives, D.C., "Supercritical Inlet Design," AIAA Paper 83-1866, 1983.
39. Mason, W.H., Rosen, B.S., "The COREL and W12SC3 Computer Programs for Supersonic Wing Design and Analysis," NASA CR 3676, 1983.
40. Moretti, G., "Considerations on Existence and Uniqueness in Numerical Inverse Design," Proceedings of the First Int. Conf. on Inverse Design Concepts in Eng. Sci., pp. 68~83, October 1984.
41. Salas, M.D., Jameson, A., and Melnik, R.E., "A Comparative study of the nonuniqueness problem of the potential Equation," AIAA Computational Fluid Dynamics Conference, 1983.
42. Giles, M.B., Drela, M., and Thompkins, Jr. W.T., "Newton Solution of Direct and Inverse Transonic Euler Equations," AIAA Paper 85-1530, 1985.
43. Giles, M.B., and Drela, M., "Two Dimensional Transonic Aerodynamic Design Method," *AIAA Journal*, Vol. 25, No. 9, pp. 1199~1206, September 1987.
44. Meauze, G., "On the Use of Inverse Modes of Calculation in Two Dimensional Cascades and Ducts," Proceedings of the First Int. Conf. on Inverse Design Concepts in Eng. Sci., October 1984.
45. Couaillier, V., and Veuillot, J.P., "Inverse Method for the Euler Equations," Proceedings of the Second Int. Conf. on Inverse Design Concepts and Optimization in Eng. Sci., pp. 131~146, October 1987.
46. Powell, A.G., Agrawal, S., and Lacey, T.R., "Feasibility and Benefits of Laminar Flow Control in Supersonic Cruise Airplanes," NASA CR-181817, July 1989.
47. Lewis, T.S., and Sirovich, L. "The Inverse Problem for Supersonic Airfoils," *AIAA Journal*, Vol.22, No.2, February 1984.
48. Fong, J., and Sirovich, L., "Direct and Inverse Problem in Supersonic Axisymmetric Flow", *AIAA Journal*, Vol.24, No.5, pp. 852~854, 1986.
49. Barger, R.L., "Procedures for Designing Supersonic Bodies of Revolution from Prescribed Surface Pressure Distributions," NASA SP-228, October 1969, pp 75-83.

50. Barger, R.L., and Brooks, Jr. C.W., "A Streamline Curvature Method for Design of Supercritical and Subcritical Airfoils," NASA TN D-7770, September 1974.
51. Campbell, R.L., and Smith, L.A., "A Hybrid Algorithm for Transonic Airfoil and Wing Design," AIAA Paper 87-2552, 1987.
52. Lighthill, M.J., "A New Method of Two-Dimensional Aerodynamic Design," R & M 2112, April 1945, Aeronautical Research Council, London, England.
53. Woods, L.C., "The Design of Two-Dimensional Aerofoils with Mixed Boundary Conditions," *J. Quart. Appl. Math.*, Vol. 13, No. 2, pp. 139~146, 1955.
54. Volpe, G., and Melnik, R.E., "The Design of Transonic Aerofoils by a Well-Posed Inverse Method," *International Journal for Numerical Methods in Engineering*, Vol. 22, pp. 341~361, 1986.
55. Volpe, G., and Melnik, R.E., "The Role of Constraints in the Inverse Design Problem for Transonic Airfoils," AIAA Paper 81-1233, June 1981.
56. Volpe, G., "The Inverse Design of Closed Airfoils in Transonic Flow," AIAA Paper 83-0504, January 1983.
57. Daripa, P., "An Exact Inverse Method for Subsonic Flows," *J. Quart. Appl. Math.*, Vol. 46, No. 3, pp. 505~526, 1988.
58. Gally, T.A., and Carlson, L.A., "Inviscid Transonic Wing Design Using Inverse Methods in Curvilinear Coordinates," AIAA Paper 87-2551, 1987.
59. Weed, R.A., Anderson, W.K., Carlson, L.A., "A Direct-Inverse Three-Dimensional Transonic Wing Design Method for Vector Computers," AIAA Paper 84-2156, August 1984.
60. Shankar, V., "Computational Transonic Inverse Procedure for Wing Design with Automatic Trailing Edge Closure," AIAA Paper 80-1390, July 1980.
61. Fink, M.R., "Hypersonic Minimum-Drag Slender Bodies of Revolution," *AIAA Journal*, Vol. 4, No. 10, pp. 1717~1724, October 1966.
62. Eggers, A.J., Jr., Resnikoff, M.M., and Dennis, D.D., "Bodies of Revolution Having Minimum Drag at High Supersonic Speeds," NACA Report 1306, 1957.

63. Parker, H.M., "Minimum Drag Ducted and Pointed Bodies of Revolution Based on Linearized Supersonic Theory," NACA R 1213, 1955.
64. Hayes, W.D., and Probstein, R.F., *Hypersonic Flow Theory*, Academic Press, New York, pp. 103~106, 1959.
65. Miele, A., "A Study of Slender Shapes of Minimum Drag Using the Newton-Busemann Pressure Coefficient Law," *AIAA Journal*, Vol. 1, No. 1, pp. 168-178, January 1963.
66. Cole, J.D., "Newtonian Flow Theory for Slender Bodies," *Journal of the Aerospace Sciences*, Vol. 24, No. 6, pp. 448-455, June 1957.
67. Zandbergen, P.J., "On the determination of optimum shapes with finite nose angles," Report NLR-TR G.30, (available as N65-17589 or N65-1549) 1964.
68. Powers, S.A., "Drag Minimization Using Exact Methods," *AIAA Journal*, Vol. 2, No. 5, pp. 941~943, May 1964.
69. Maestrello, L., and Ting, L., "Optimum Shape of a Blunt Forebody in Hypersonic Flow," ICASE Report No. 89-51, December 1989.
70. Stivers, L.B., Jr., and Spencer, B., Jr., "Studies of Optimum Body Shapes," in *Conference on Hypersonic Aircraft Technology*, NASA SP-148, May 1967.
71. Perkins, E.W., Jorgensen, L.H., and Sommer, S.C., "Investigation of the Drag of Various Axially Symmetric Nose Shapes of Fineness Ratio 3 for Mach Numbers from 1.24 to 7.4," NACA R 1386, 1958.
72. Spencer, B, Jr., and Fox, C.H., Jr., "Hypersonic Aerodynamic Performance of Minimum-Wave-Drag Bodies," NASA TR R-250, 1966.
73. Fournier, R.H., Spencer, B., Jr., and Corlett, W.A., "Supersonic Aerodynamic Characteristics of a Series of Related Bodies with Cross-Sectional Ellipticity," NASA TN D-3539, 1966.
74. Thomas, J.L., van Leer, B., and Walters, R.W., "Implicit Flux-Split Schemes for the Euler Equations," AIAA Paper 85-1680, July 1985.

75. Grossman, B., and Walters, R.W., "An Analysis of Flux-Split Algorithms for Euler's Equations with Real Gases," AIAA Paper No. 87-1117, June 1987.
76. Walters, R.W., Cinnella, P, Slack, D.C., and Halt, D., "Characteristic-Based Algorithms for Flows in Thermo-Chemical Nonequilibrium," AIAA Paper No. 90-0393, January 1990.
77. Ashley, H., and Landahl, M., *Aerodynamics of Wings and Bodies*, Addison-Wesley, 1965, pp. 178~181.
78. Mason, W.H., *Aerodynamic Calculation Methods Pak # 2 Basic Geometry for Aerodynamics*, Aerocal, 1981, p. 27.
79. Krasnov, N.F., *Aerodynamics of Bodies of Revolution*, American Elsevier Publishing Company Inc, 1970, pp.545~548, pp. 463~470, pp. 163~184.
80. Liepmann, H.W., and Roshko, A., *Elements of Gas Dynamics*, John Wiley and Sons, New York, 1957, pp. 212~215.
81. Carafoli, E., *Wing Theory in Supersonic Flow*, Pergamon Press, 1969, pp. 463~473.
82. Cox, R.N., and Crabtree, L.F., *Elements of Hypersonic Aerodynamics*, Academic Press, New York, 1965 pp. 97~100.
83. Cruz, C.I., "Improved Tangent-Cone Method for the Aerodynamic Preliminary Analysis System Version of the Hypersonic Arbitrary-Body Program," NASA TM 4165, 1990.
84. Eggers, Jr., A.J., and Savin, R.C., "A Unified Two-Dimensional Approach to the Calculation of Three-Dimensional Hypersonic Flows, with Application to the Bodies of Revolution," NACA Report 1249, 1952.
85. Handbook of Supersonic Aerodynamics, Section 8 Bodies of Revolution, NAVWEPS Report 1488 (Volume 3), October 1961.
86. Lee, Jaewoo and Mason, W.H., "Development of an Efficient Inverse Method for Supersonic and Hypersonic Body Design," AIAA Paper 91-0395, January 1991.

87. Chapkis, R.L., "Hypersonic Flow Over an Elliptic Cone: Theory and Experiment," *J. Aero. Sci.*, Vol 28, pp. 844-864, 1961.
88. Siclari, M.J. and Rubel, A., "Entropy Corrections to Supersonic Conical Nonlinear Potential Flows," *Computers & Fluids*, Volume 13, No. 3, pp. 337~359, 1985.
89. Mason, W.H., "SC³ - A Wing Concept for Supersonic Maneuvering," AIAA Paper, 83-1858, July 1983.
90. Lee, Jaewoo, and Mason, W.H., "A Three Dimensional Inverse Method for Supersonic and Hypersonic Body Design," Submitted to the AIAA Applied Aerodynamics Conference, September 1991.
91. Thompson, J.F., Warsi, Z.U.A., Mastin, C.W., *Numerical Grid Generation*, North-Holland, 1985, pp 310 ~315.

FIGURES

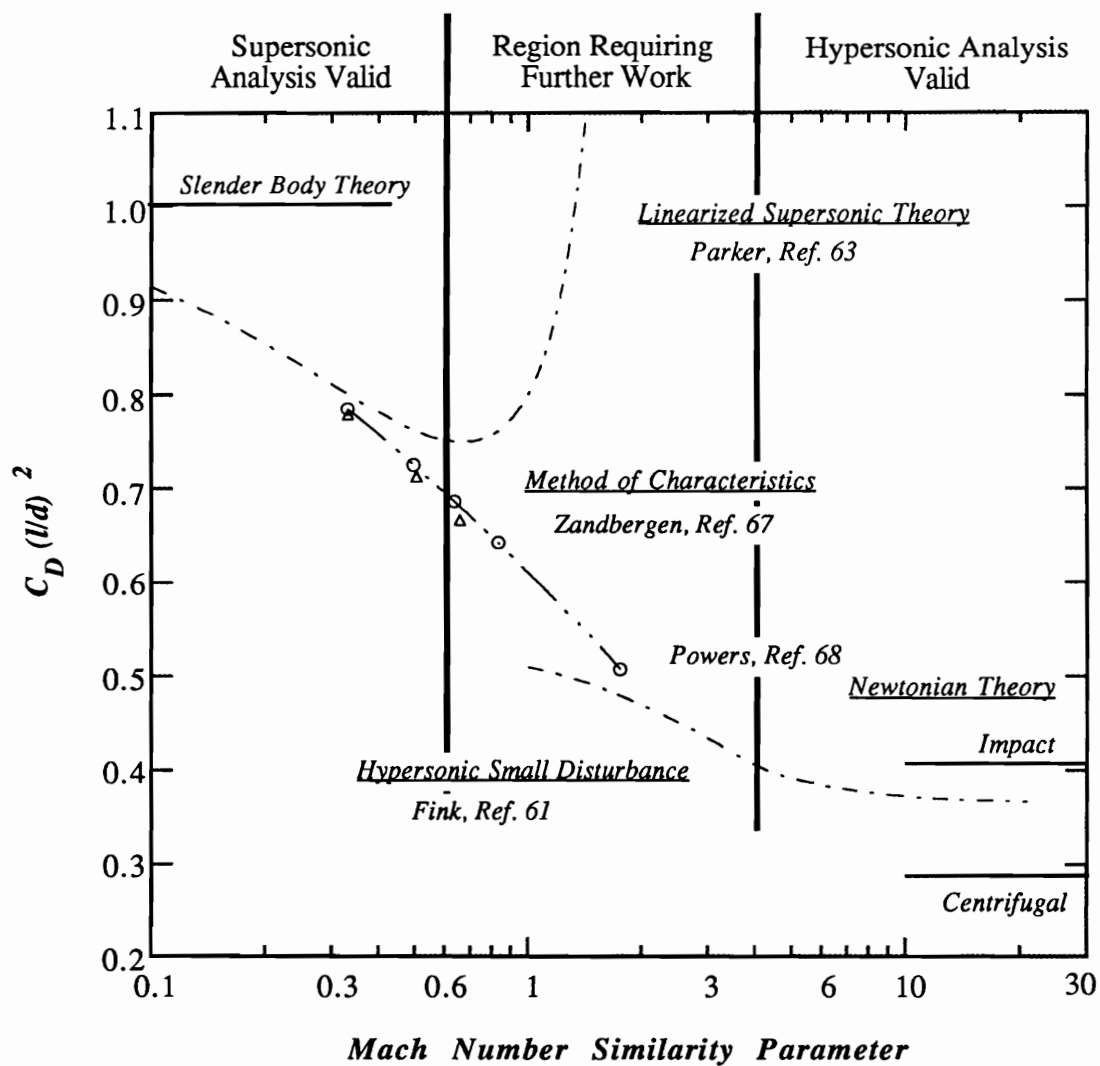


Figure 1. Demonstration of “Gap” between Linearized Supersonic and Hypersonic Minimum Drag Body Theory

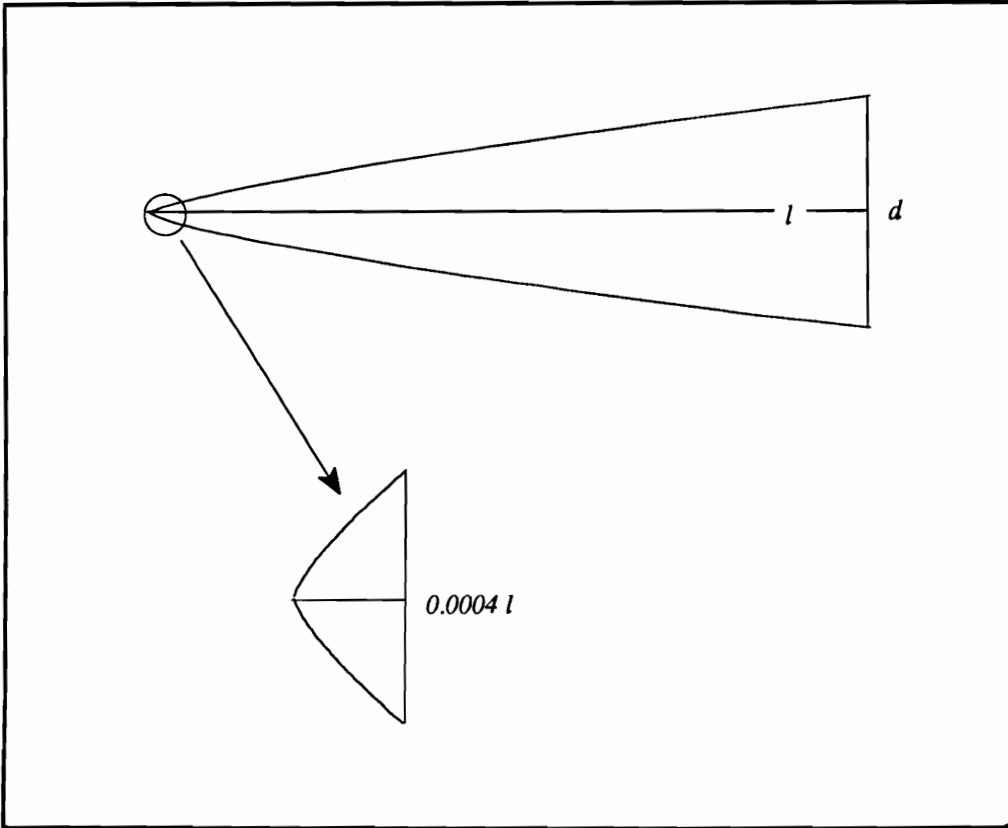


Figure 2. Details of Blunt Nose Shape

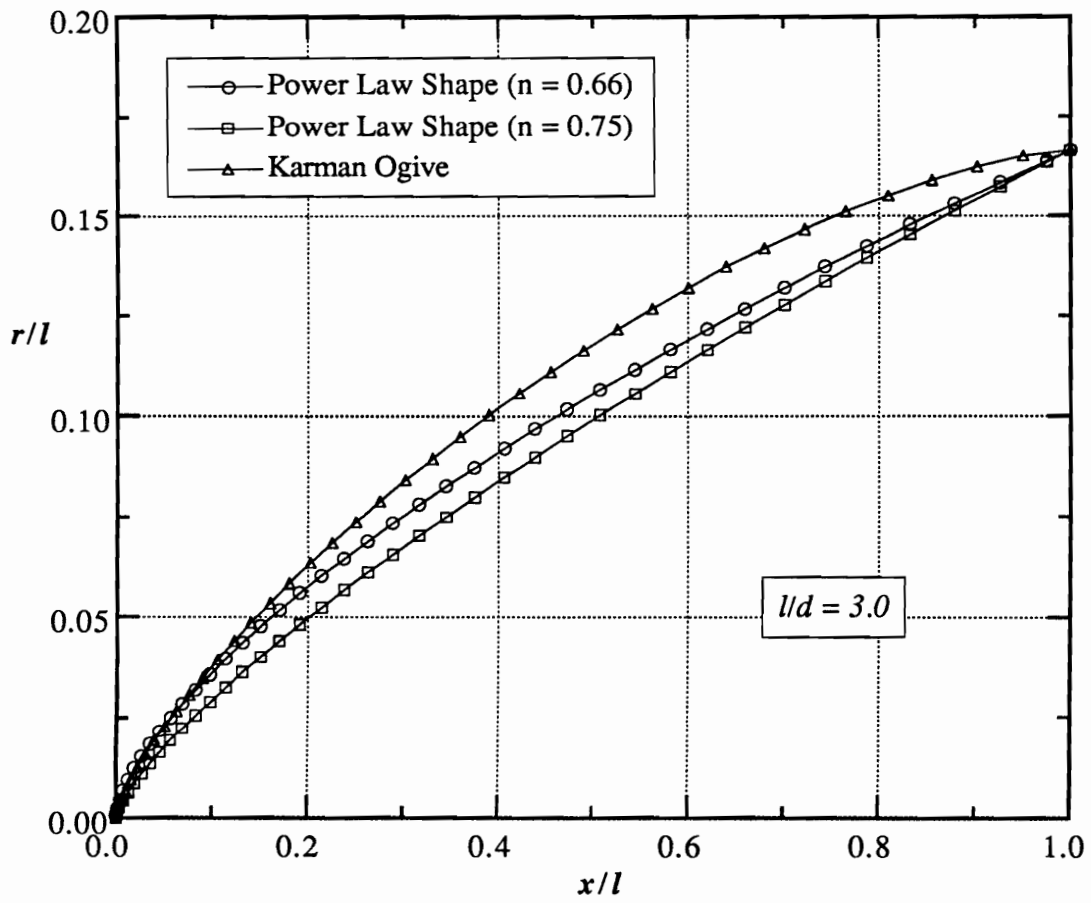
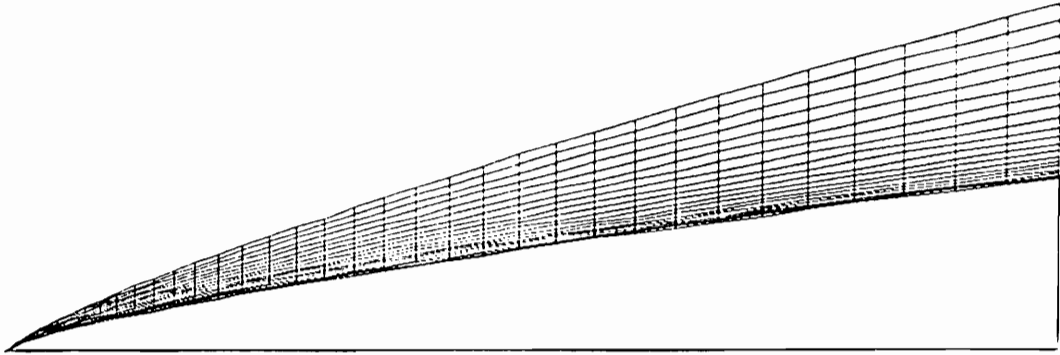
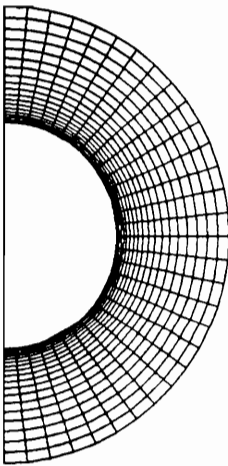


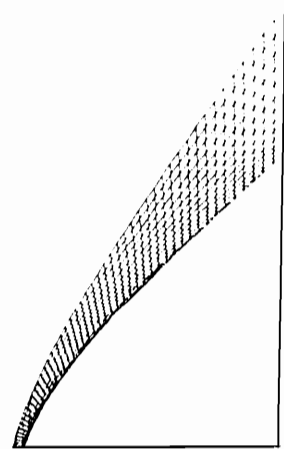
Figure 3. Comparison of Minimum Drag Shapes



(a) Axial Grid



(b) Cross Plane Grid



(c) Axial Grid near the Nose

Figure 4. Computational Grid Used in the Minimum Drag Body Study
 ($n=0.75$, $M_{\infty}=6.28$, $l/d=3.0$)

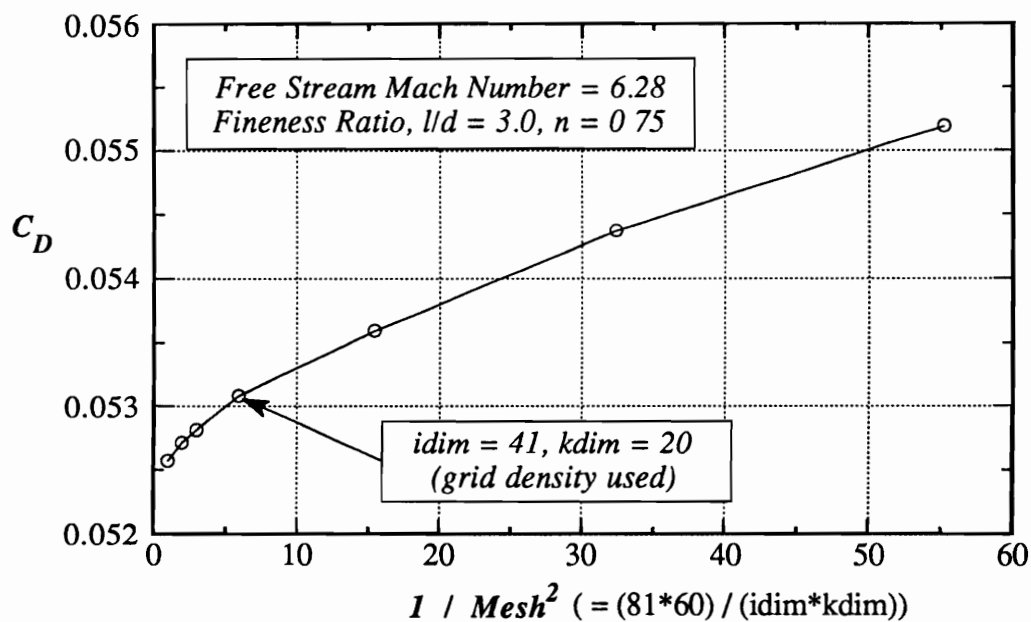


Figure 5. Convergence of Drag with Increasing Grid Density

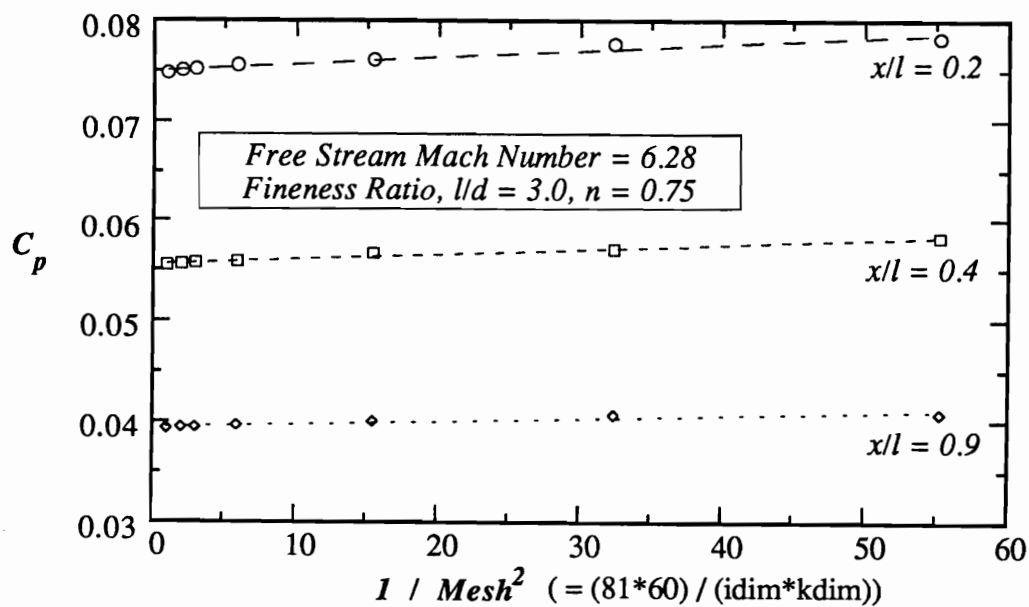
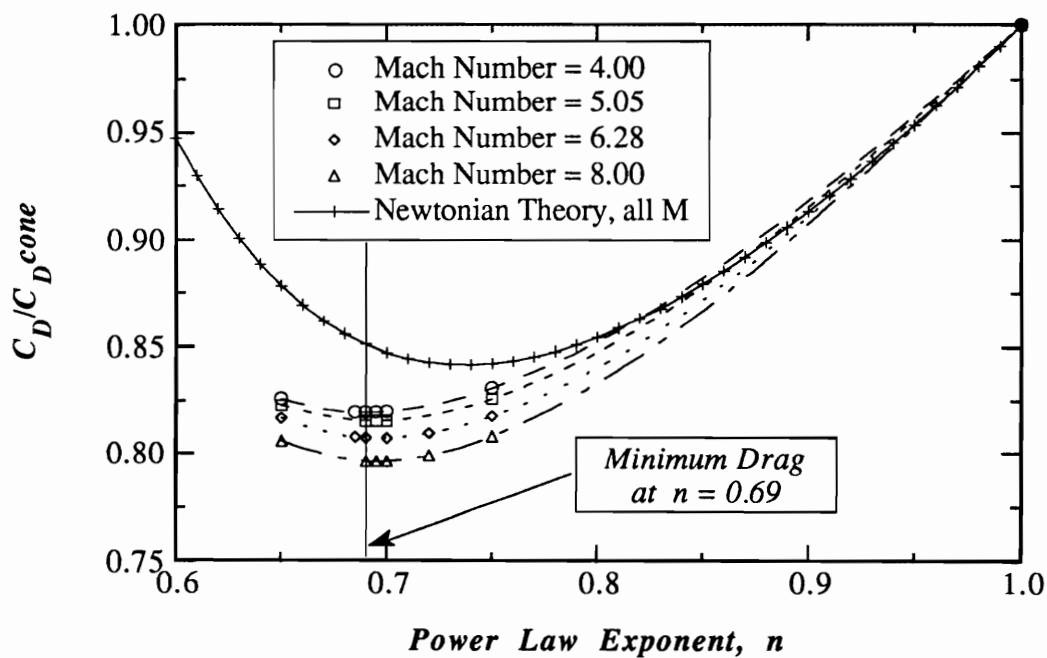
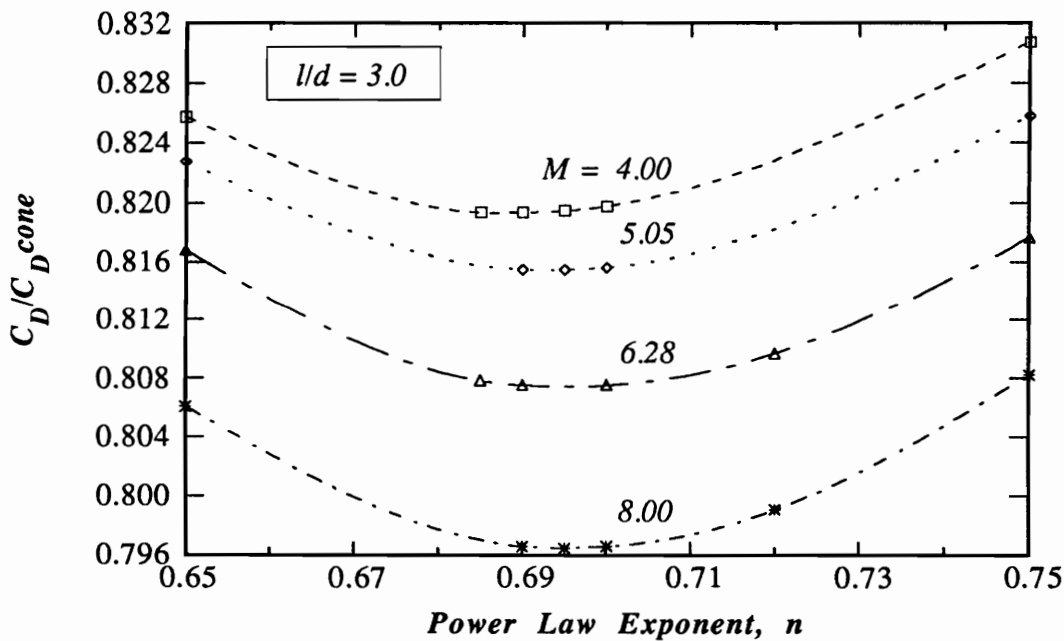


Figure 6. Convergence of Pressure Distribution with Increasing Grid Density



(a) Comparison with Newtonian Theory



(b) Examination of Detailed Behavior

Figure 7. Numerical Determination of Optimum "n" for Power Law Shapes

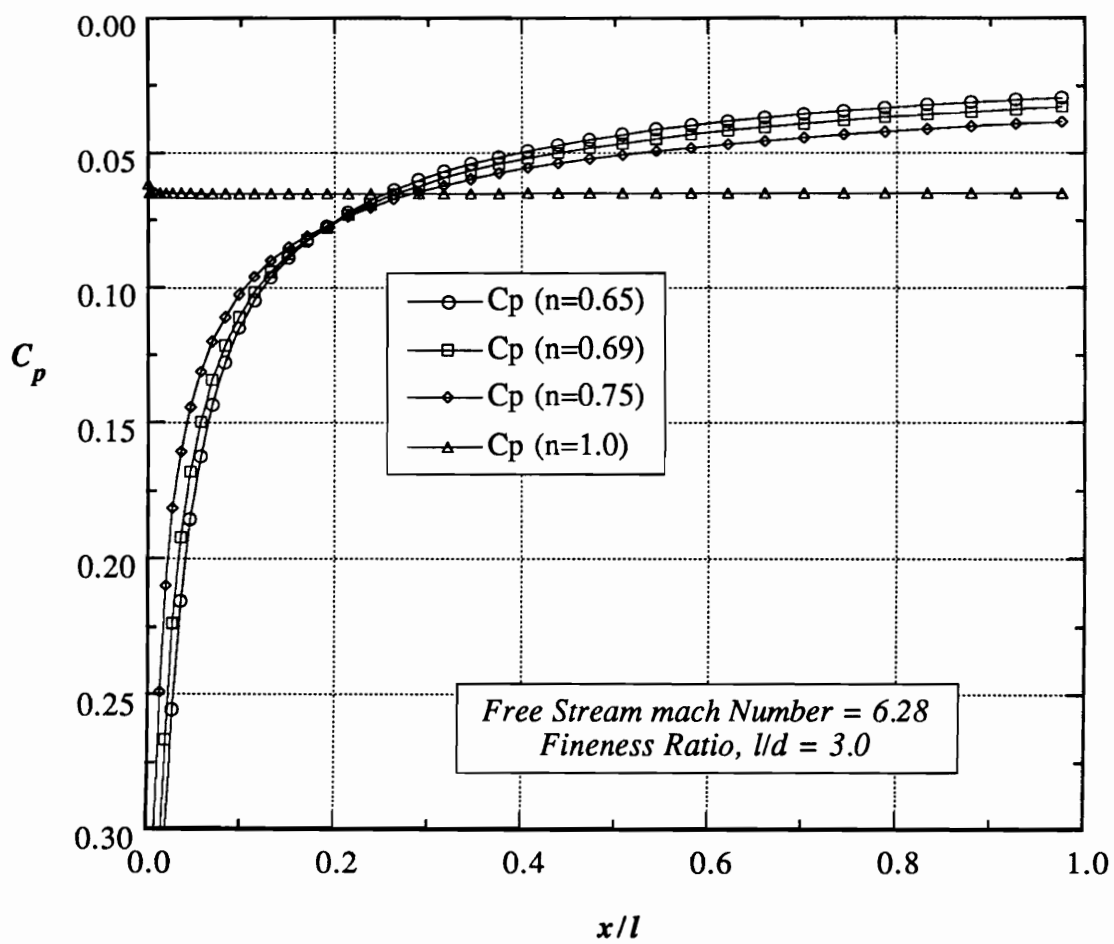


Figure 8. Pressure Distributions for Several Power Law Shapes

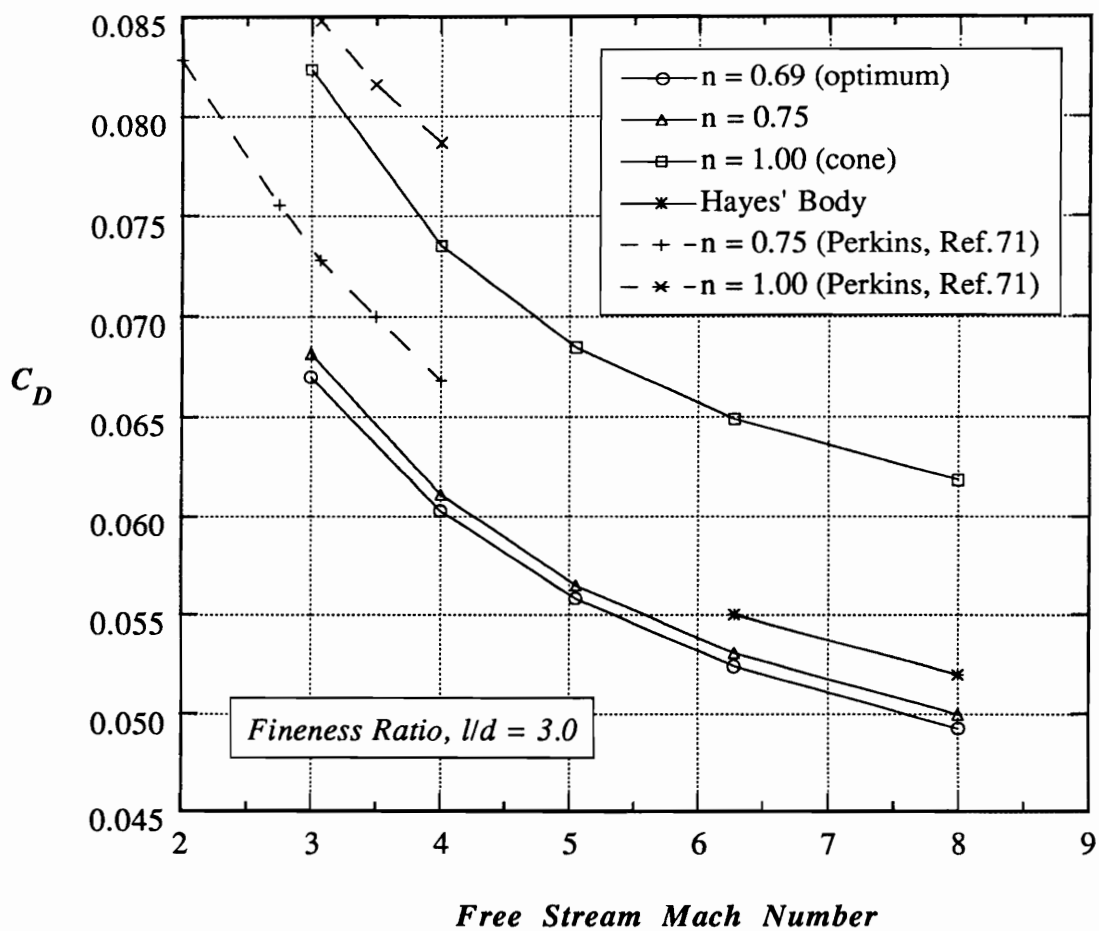


Figure 9. Mach Number Effects for Several Shapes

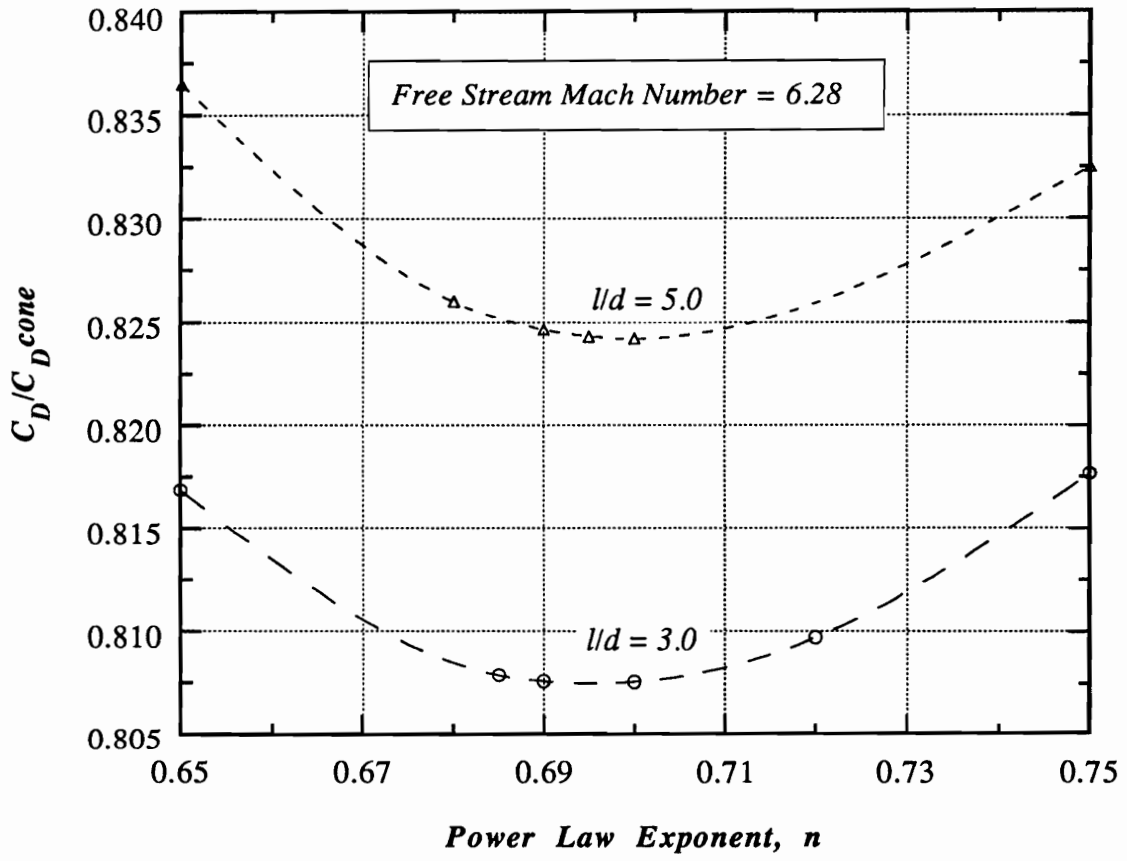


Figure 10. Comparison of $l/d = 3.0$ & 5.0

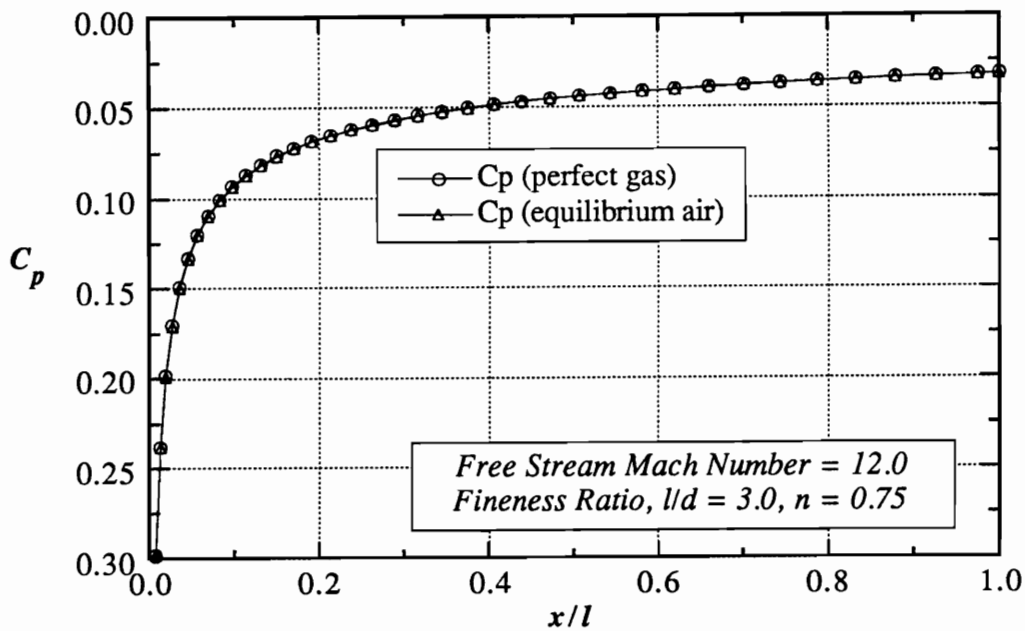


Figure 11. Comparison of Pressure Distributions of Perfect Gas and Equilibrium Air

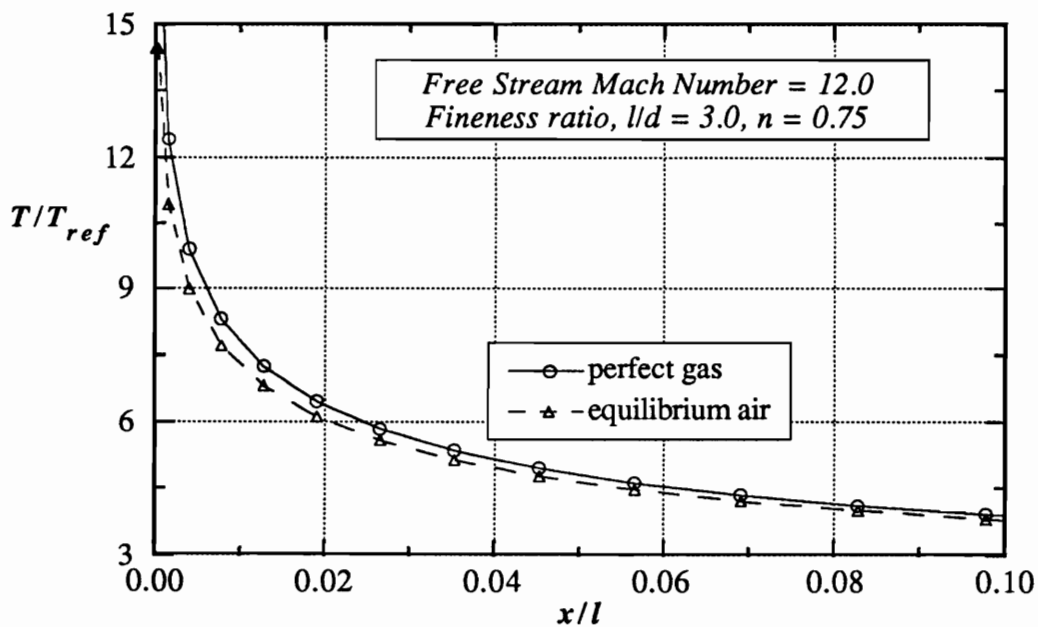


Figure 12. Comparison of Temperature Distributions between
Perfect Gas and Equilibrium Air

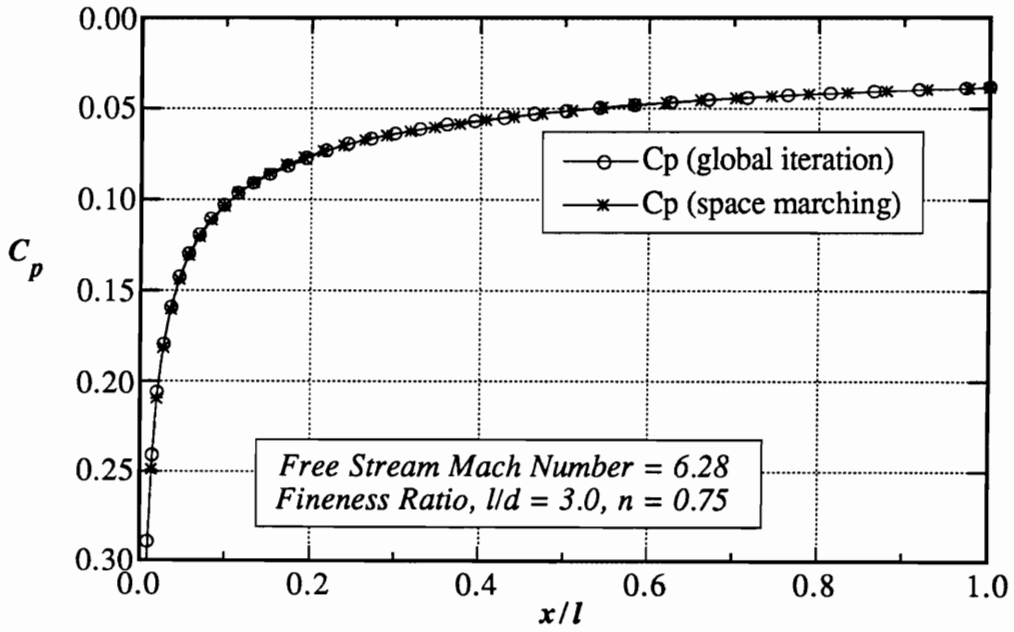


Figure 13. Comparison of Pressure Distributions for Different Numerical Schemes

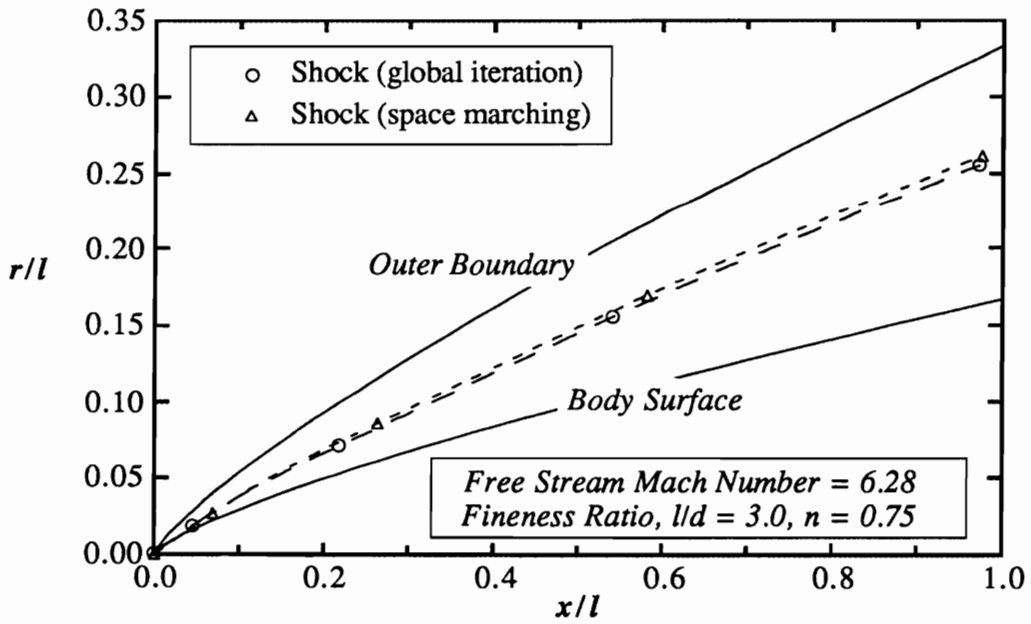


Figure 14. Comparison of the Shock Locations for the Different Numerical Schemes

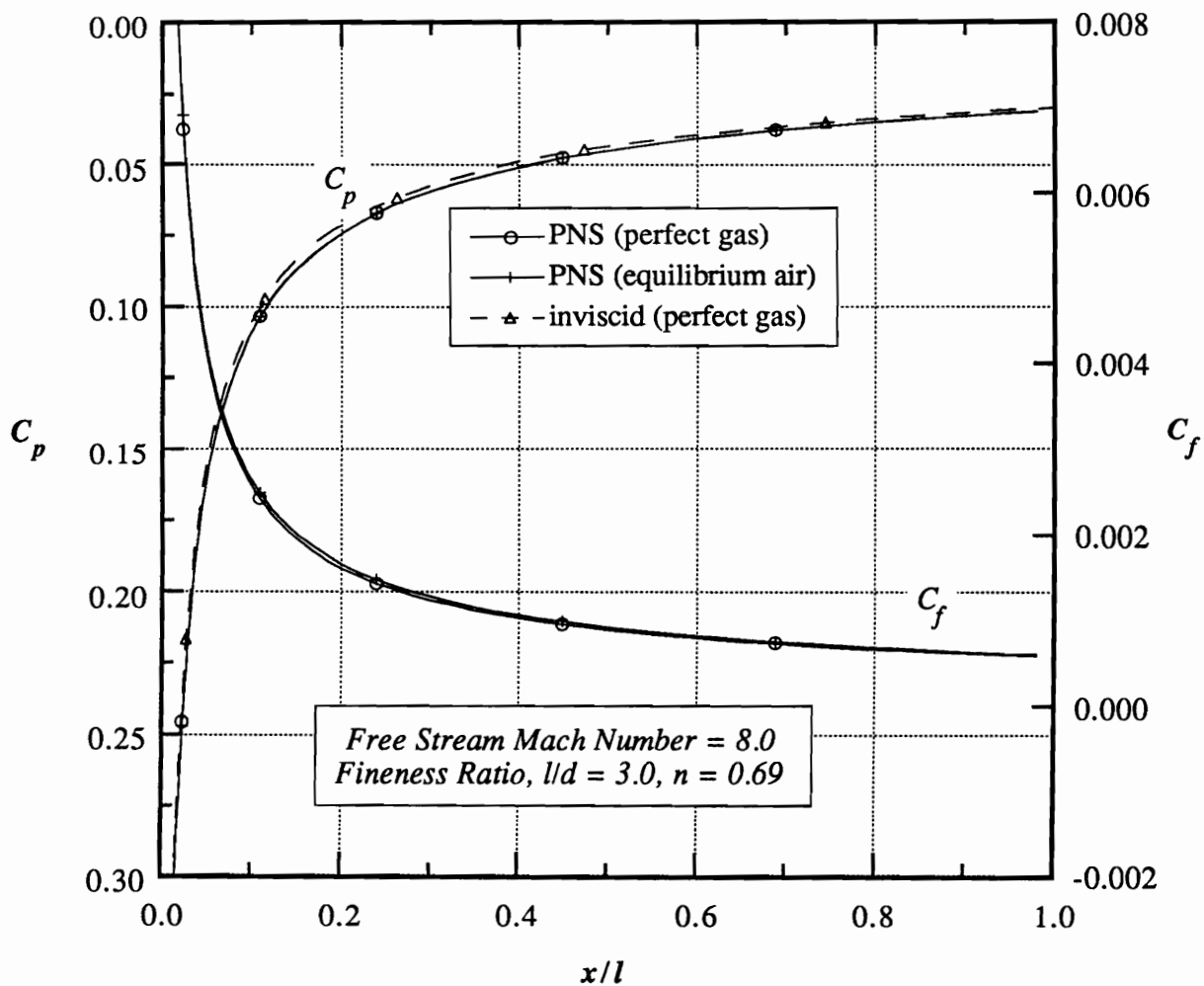


Figure 15. Pressure and Skin Friction Distribution for the Optimum Power Law Shape
(Adiabatic Wall Boundary Condition for PNS Calculation)

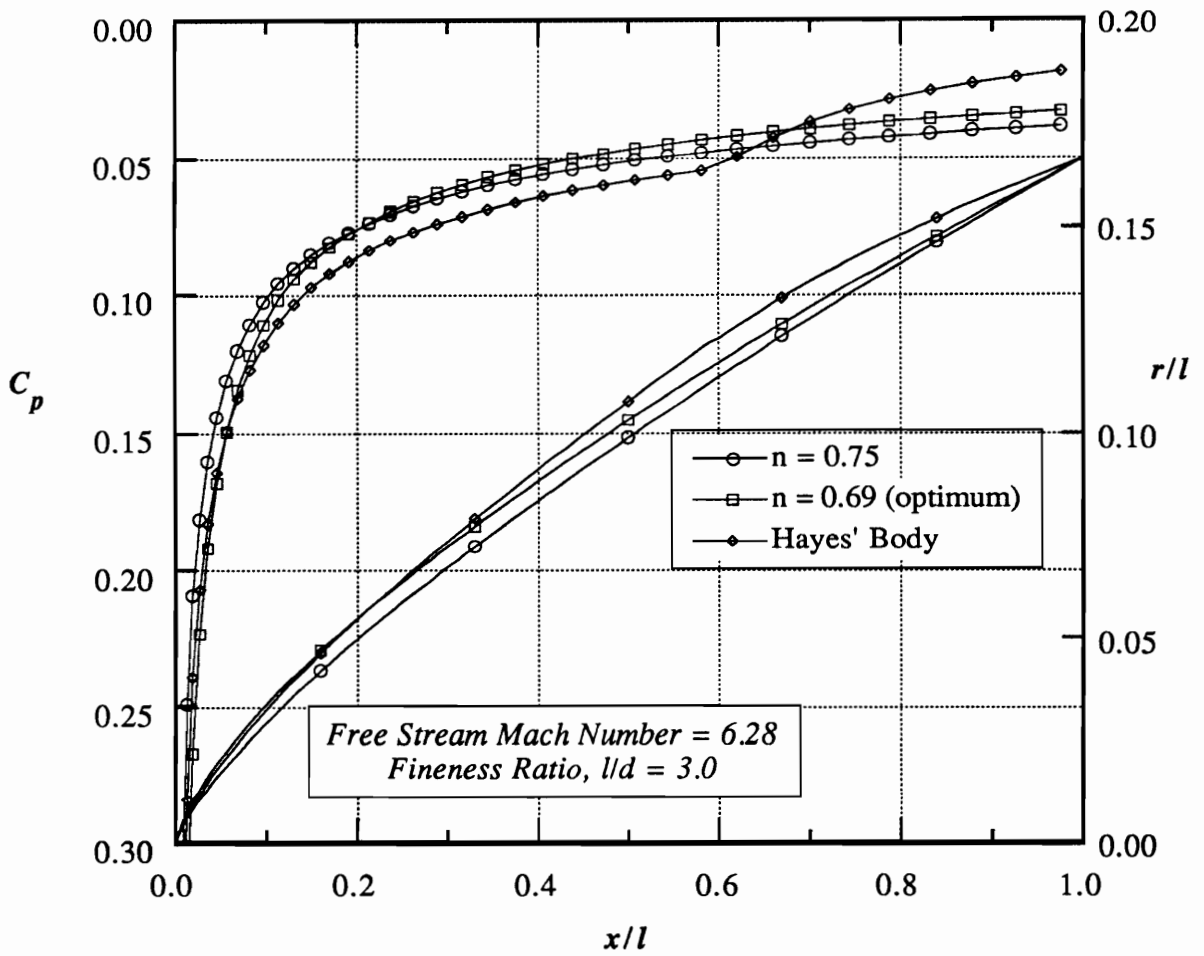


Figure 16. Comparison of Power Law Shapes and Pressure Distributions with Hayes' Body Shape

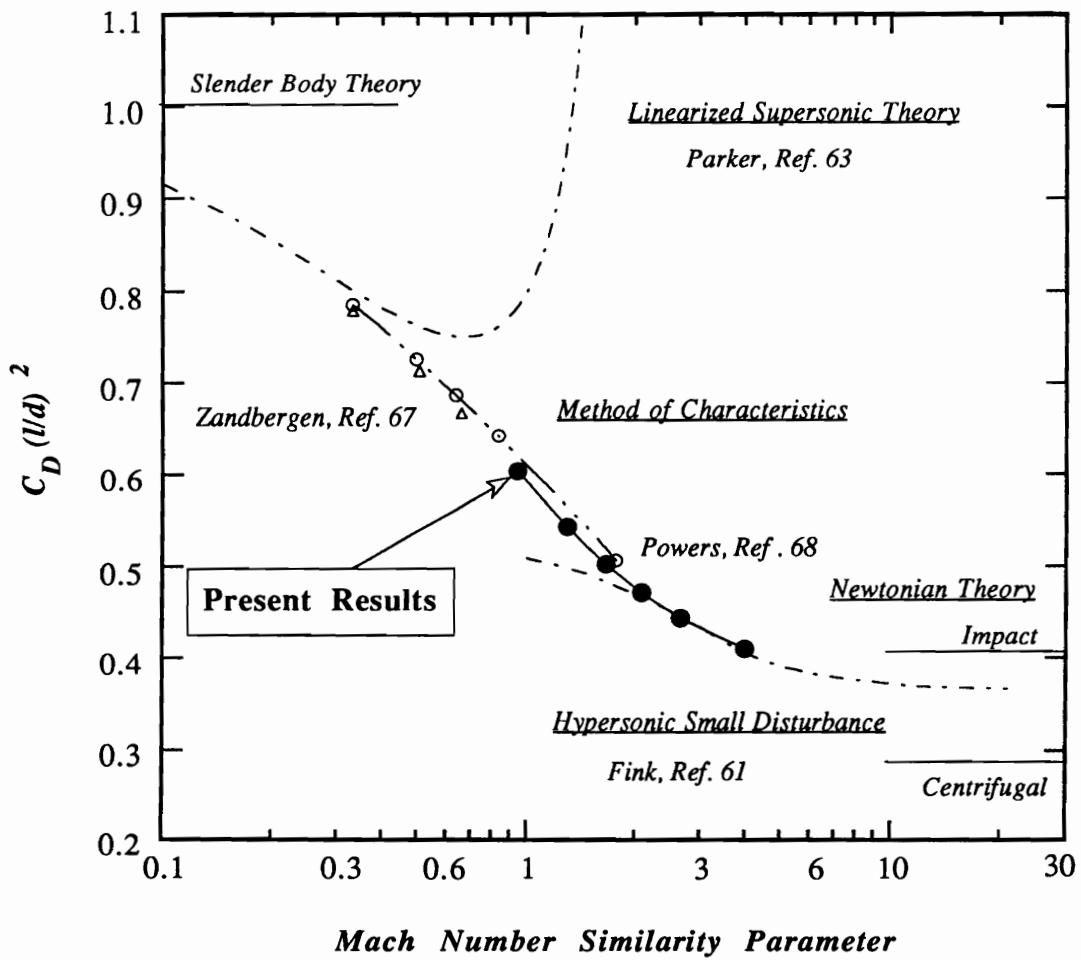


Figure 17. Results of Minimum Drag Body Study Added to the Fink⁶¹ Chart

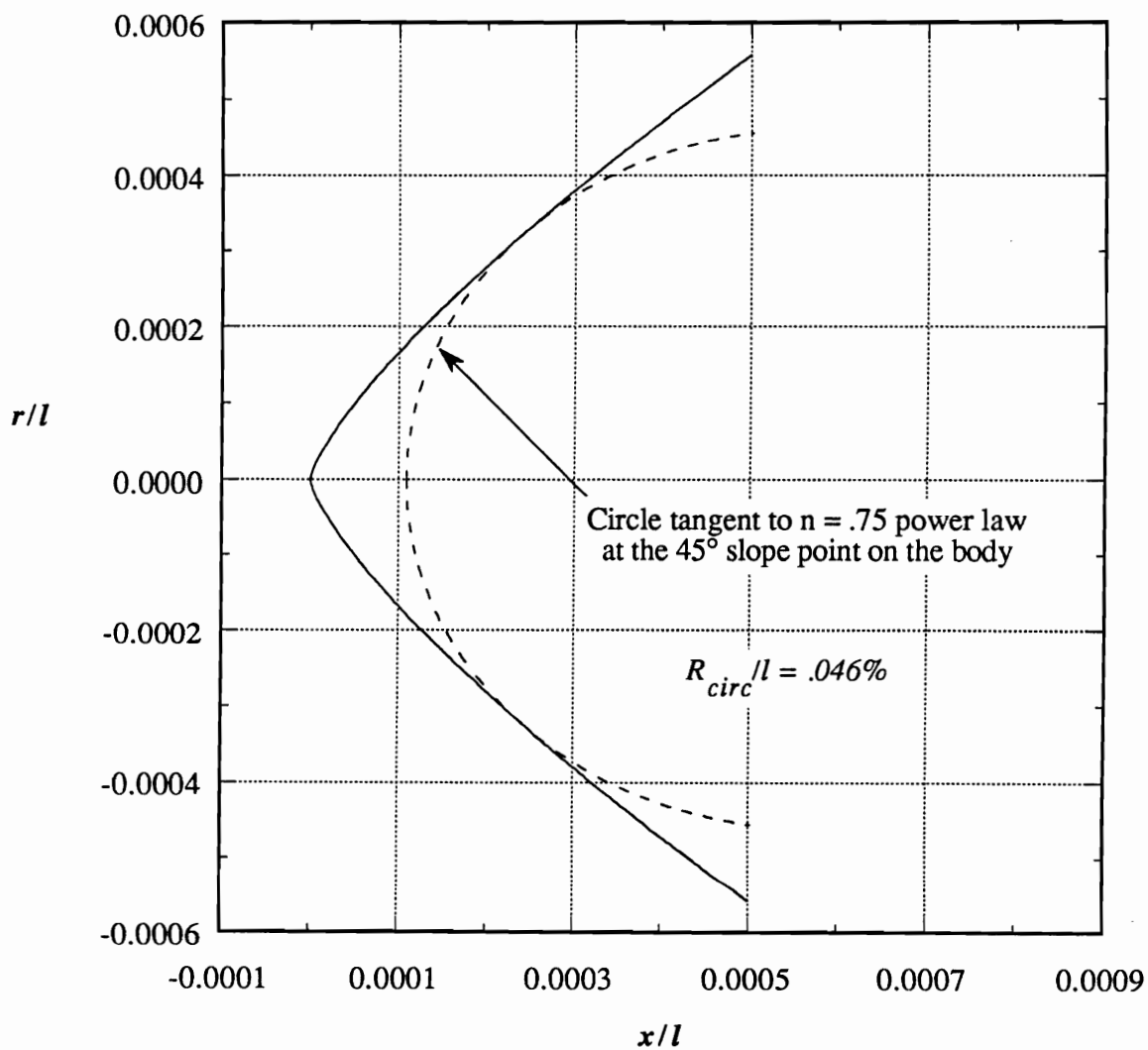


Figure 18. Comparison of Power Law Body ($n=0.75$) with Inscribed Circle near the Nose

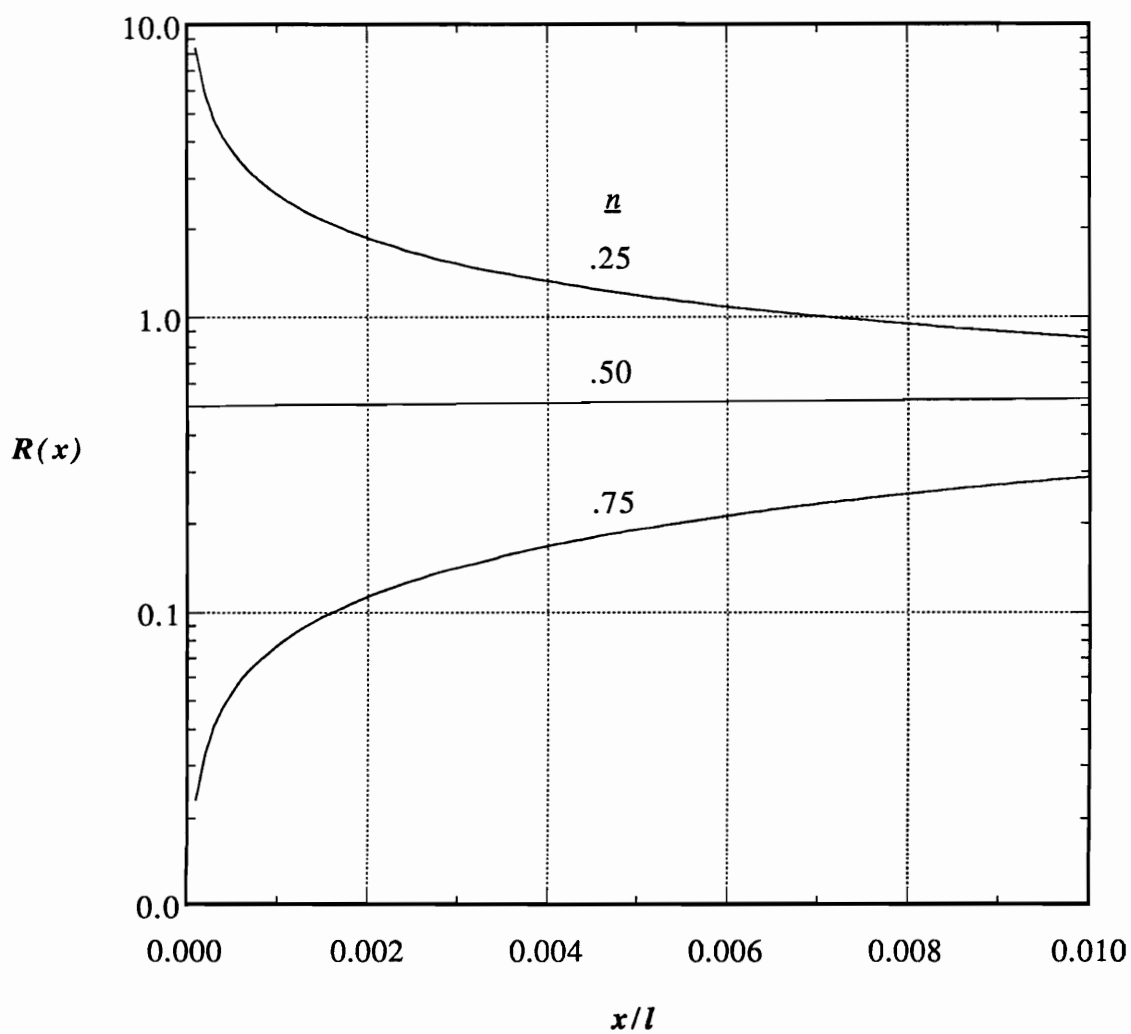


Figure 19. Curvature Distribution of Various Power Law Bodies near the Nose

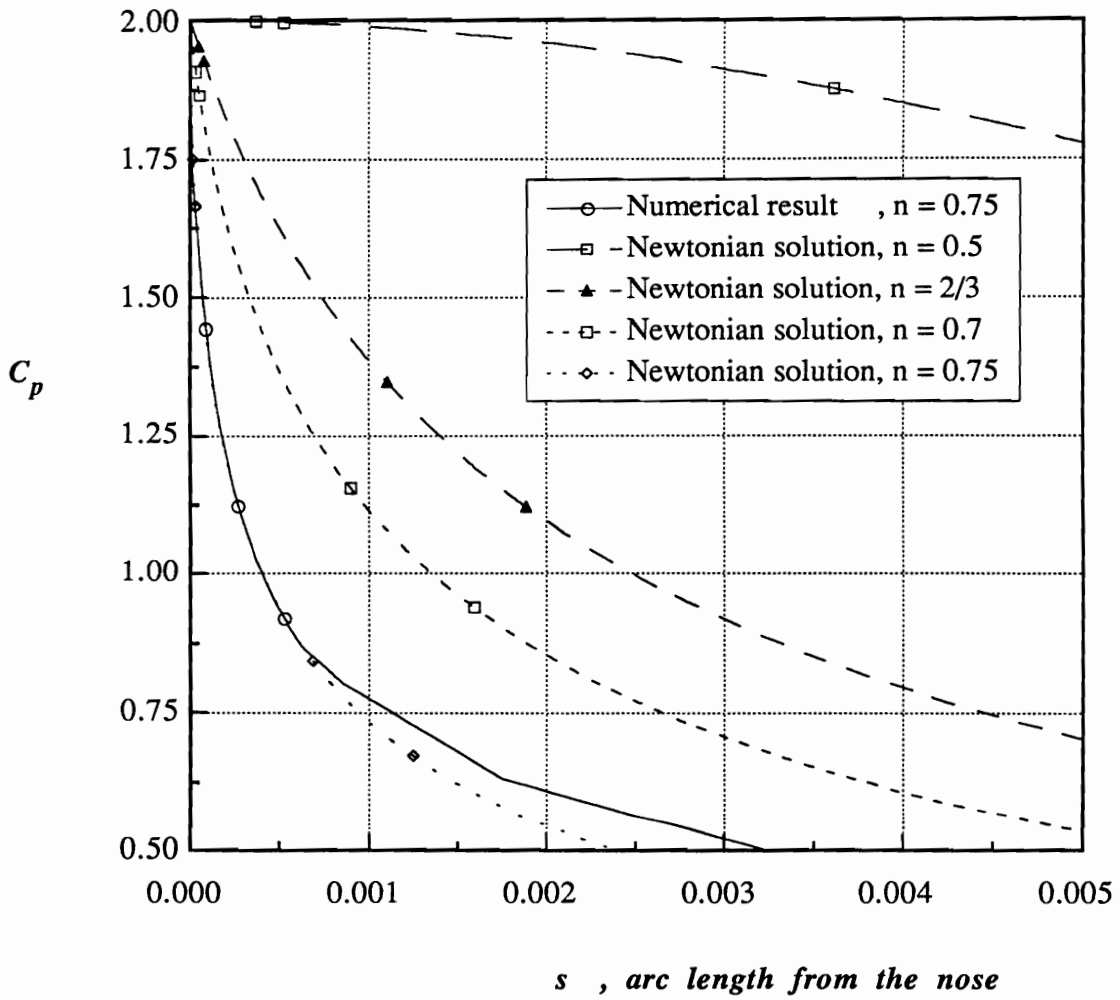


Figure 20. Newtonian Solutions and Numerical Result for Power Law Bodies; C_p vs. s
 - note dramatic effect of $n > 1/2$ -

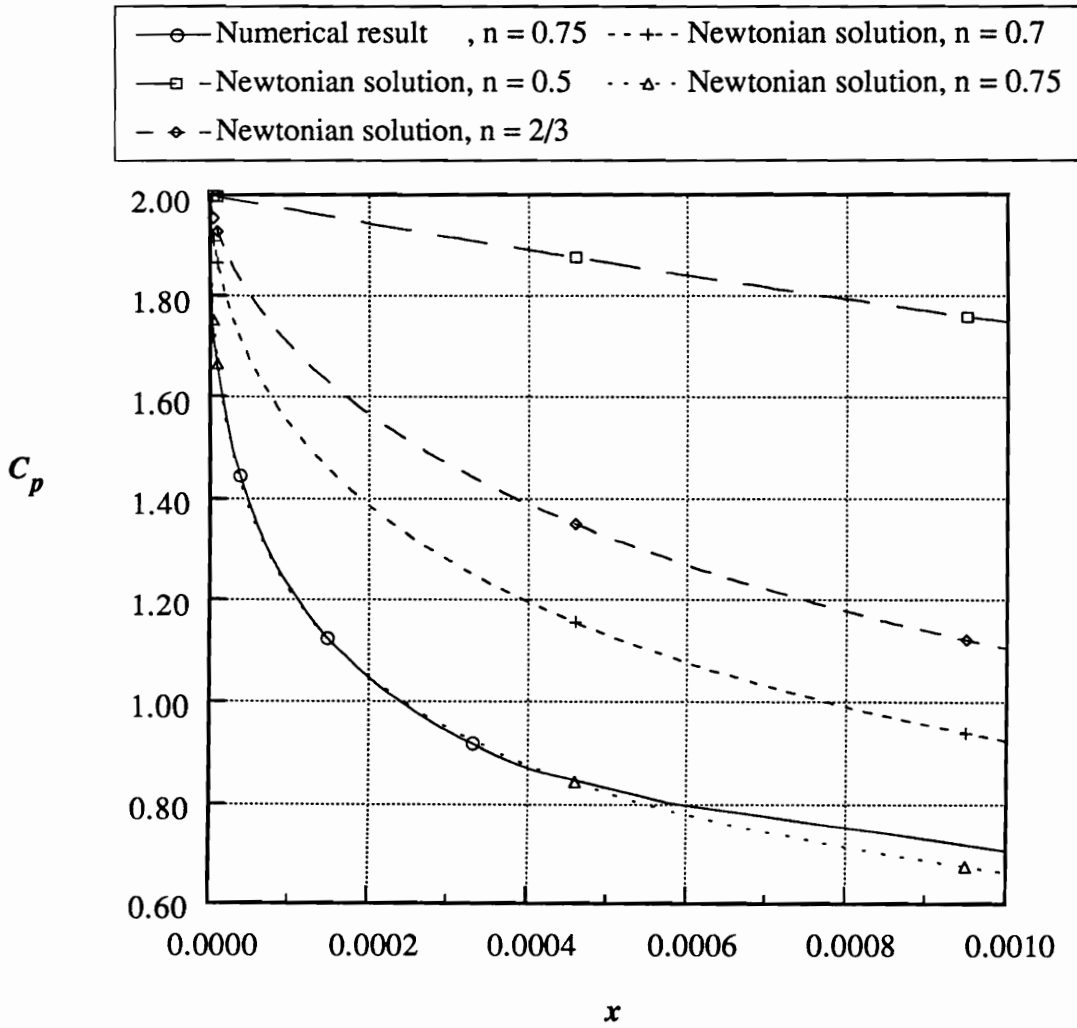


Figure 21. Newtonian Solutions and Numerical Result for Power Law Bodies; C_p vs. x
 - note dramatic effect of $n > 1/2$ -

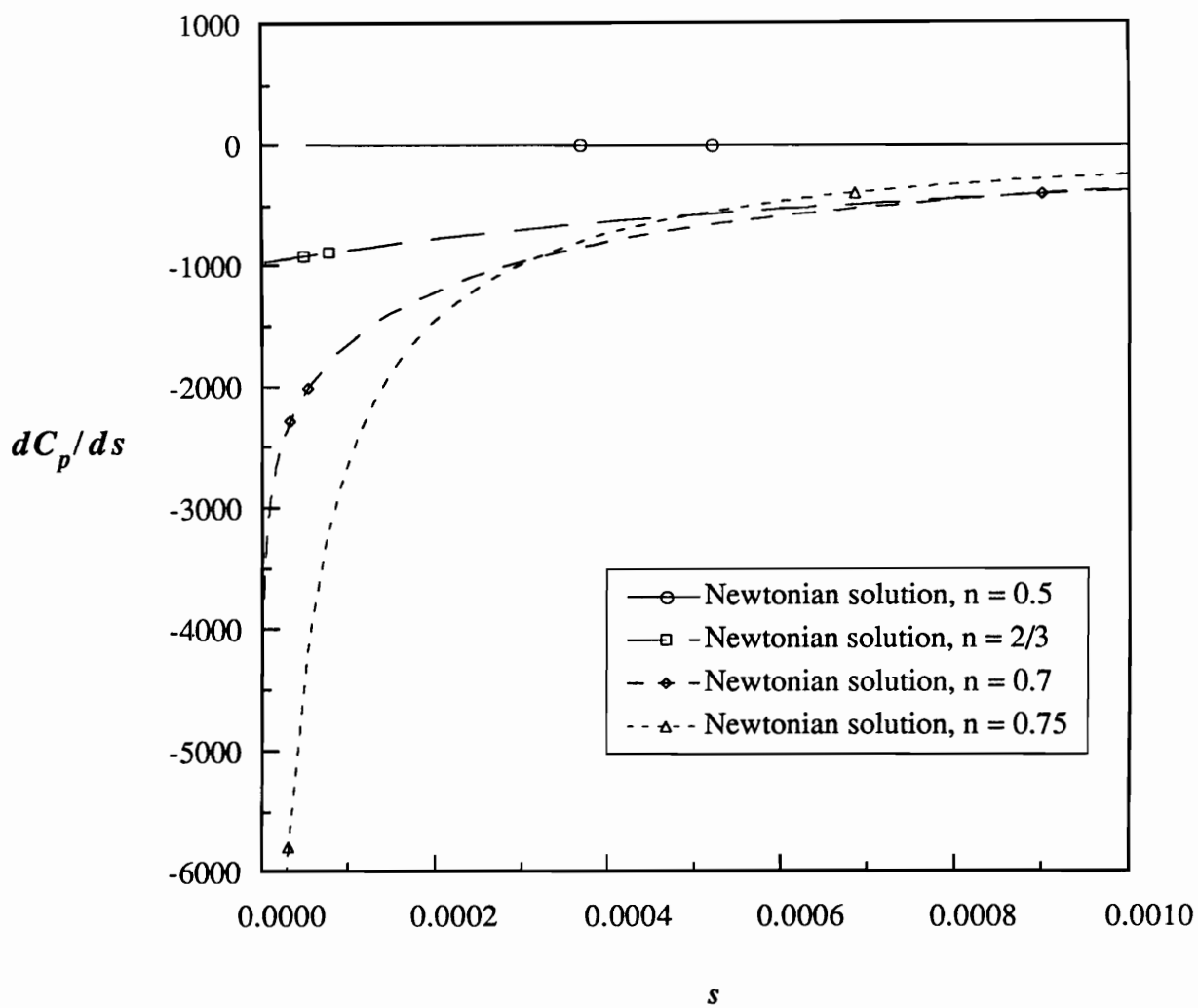


Figure 22. Newtonian Solutions for Power Law Bodies; dC_p/ds vs. s
 - note dramatic effect of $n > 1/2$ -

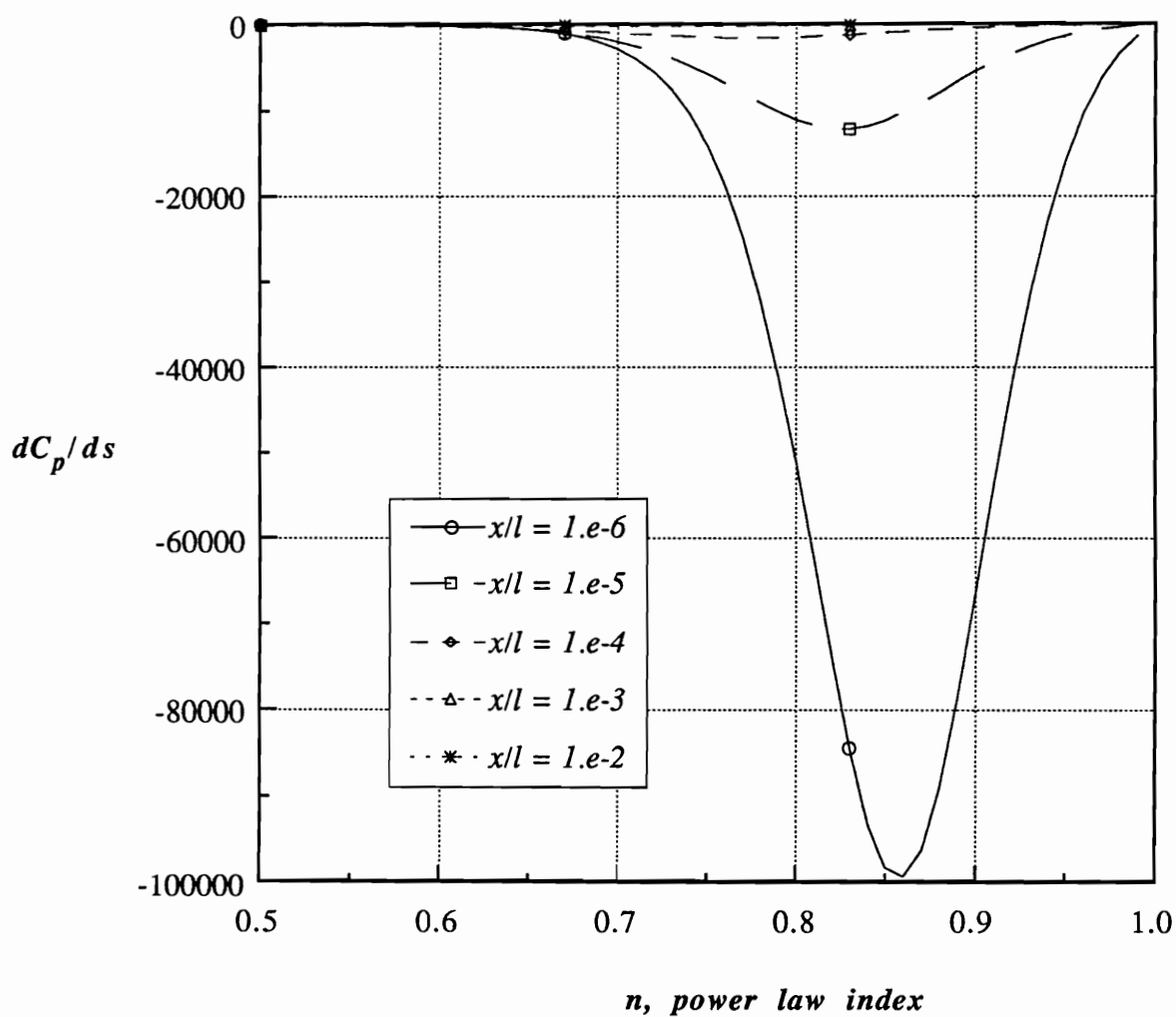


Figure 23. Newtonian Solutions for Power Law Bodies; dC_p/ds vs. power n
 - note dramatic effect of $n > 1/2$ -

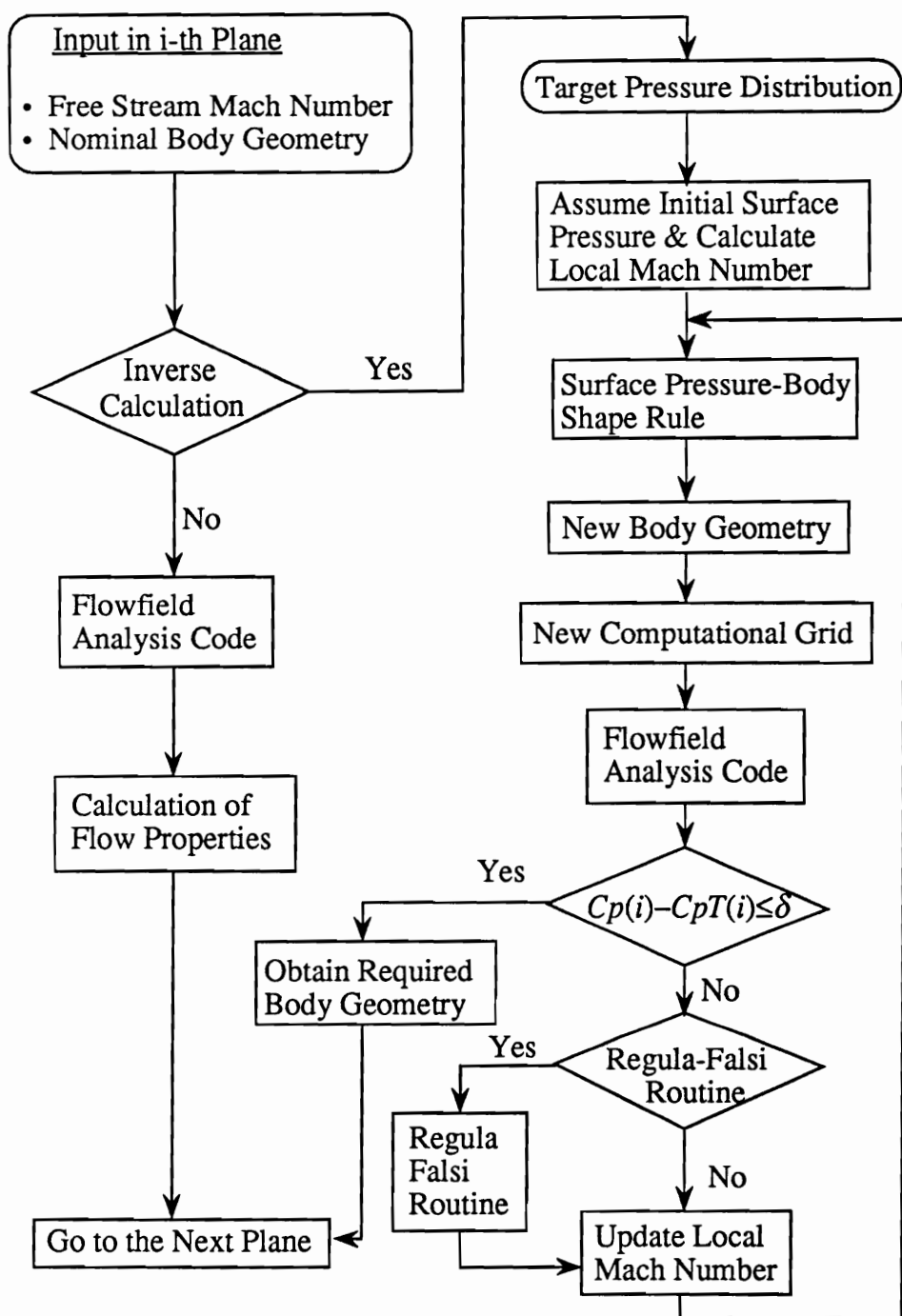


Figure 24. Flow Chart of Inverse Method for Axisymmetric Bodies

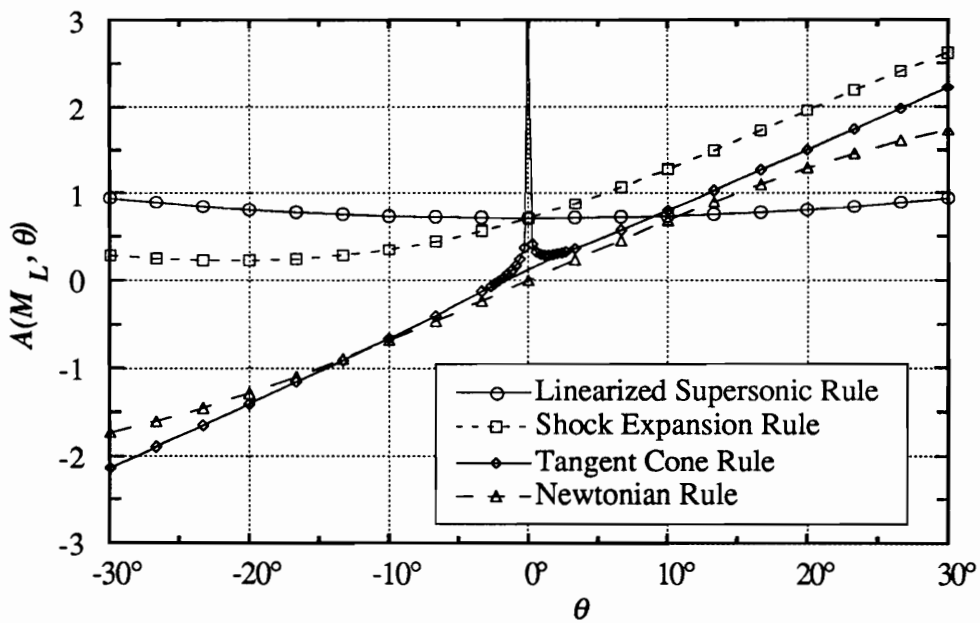


Figure 25. Comparison of $A(M_L, \theta)$ in Four Pressure-Shape Rules ($M_\infty = 3.0$)

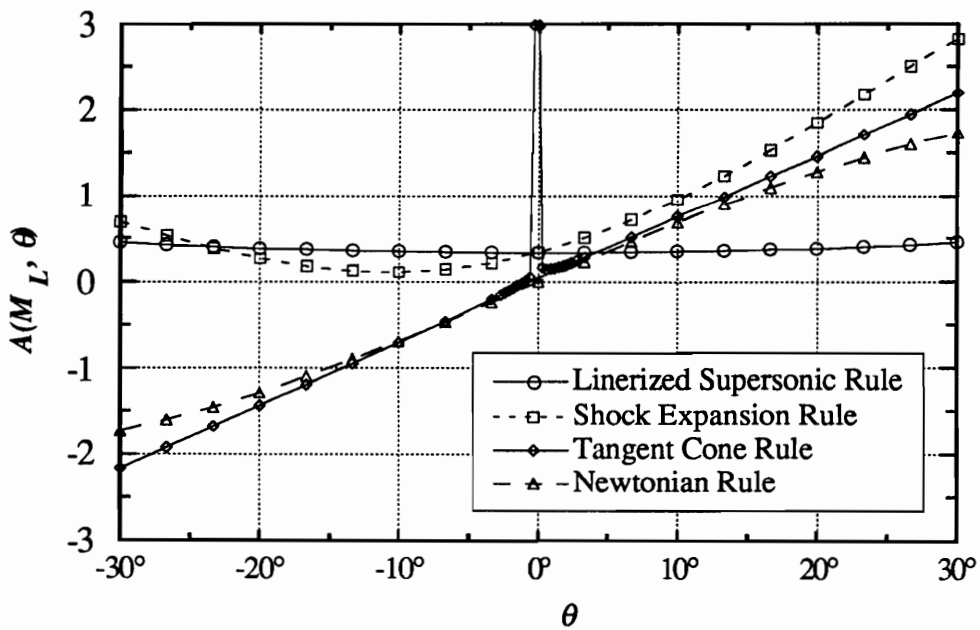
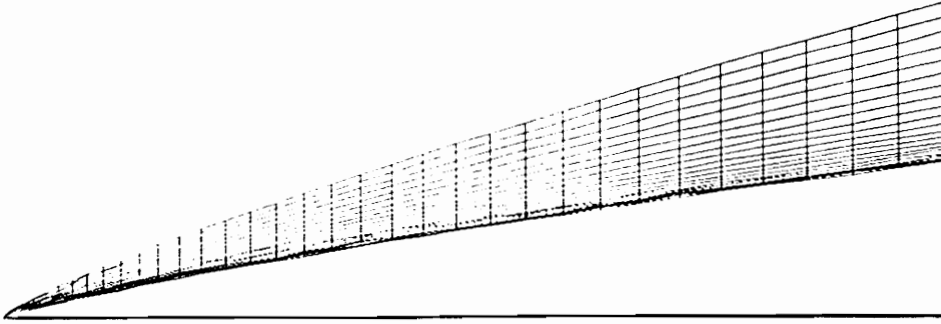
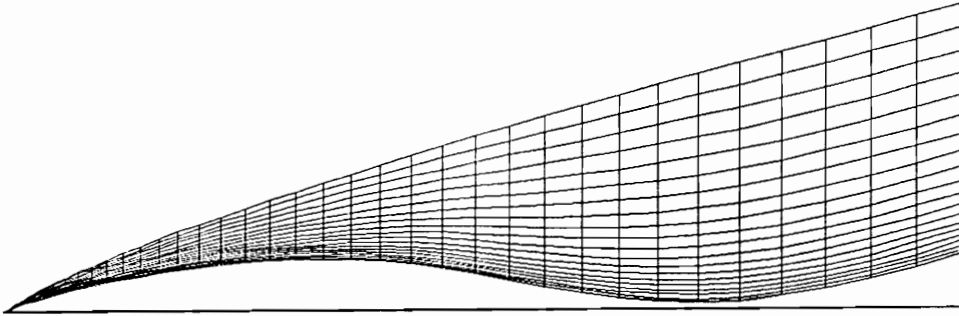


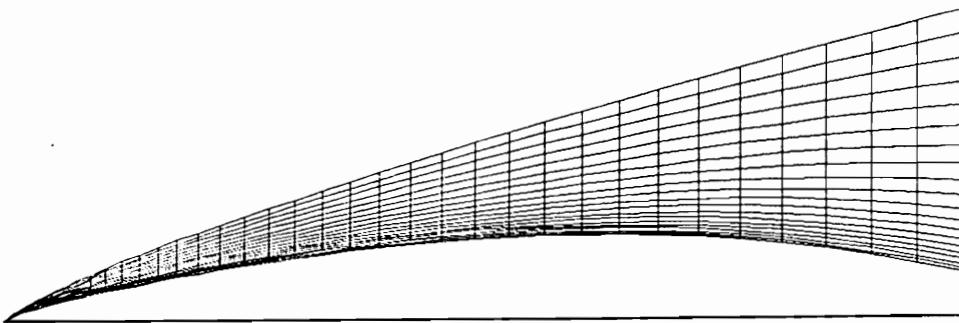
Figure 26. Comparison of $A(M_L, \theta)$ in Four Pressure-Shape Rules ($M_\infty = 6.0$)



(a) Nominal Body (3/4 Power Law Shape)



(b) Test Case I



(c) Test Case II

Figure 27. Axial Computational Grids for Axisymmetric Inverse Test Cases

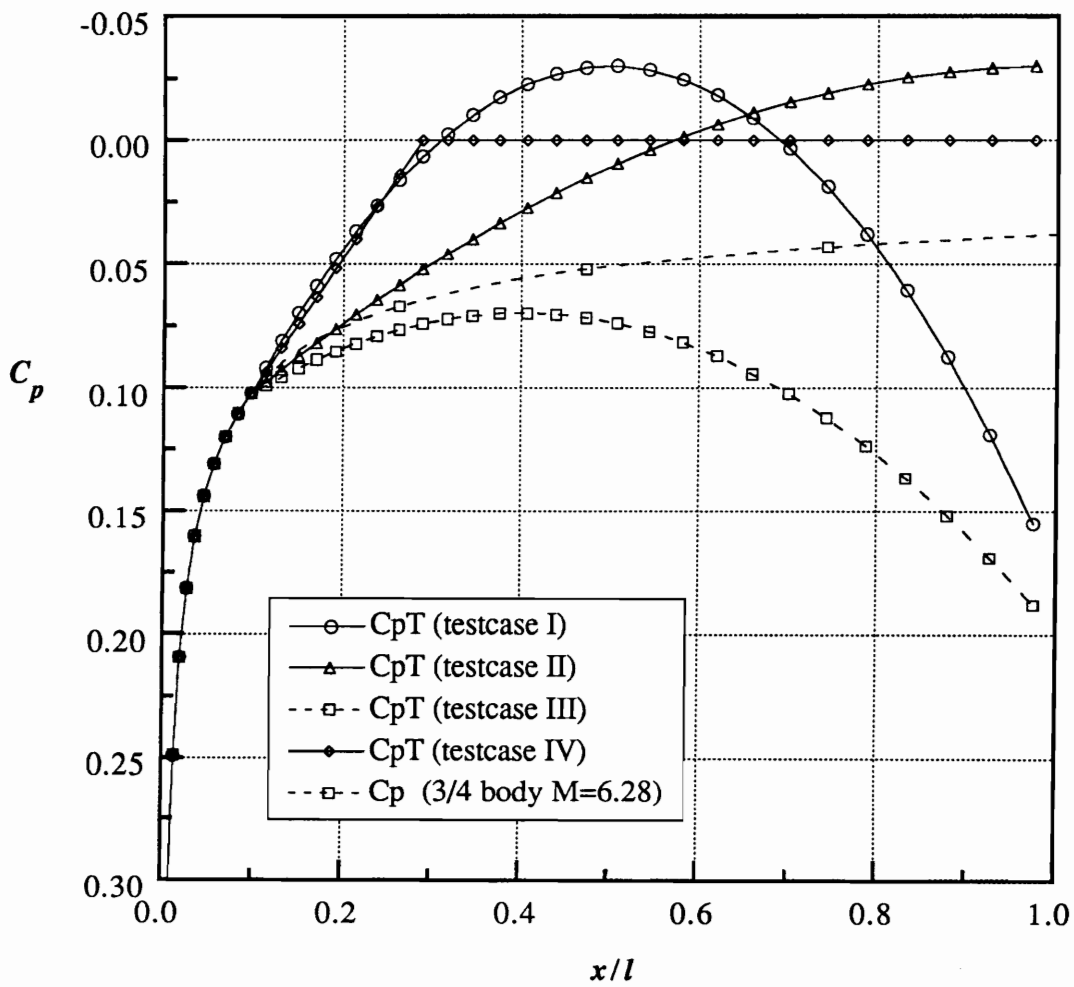


Figure 28. Test Target Pressure Distributions

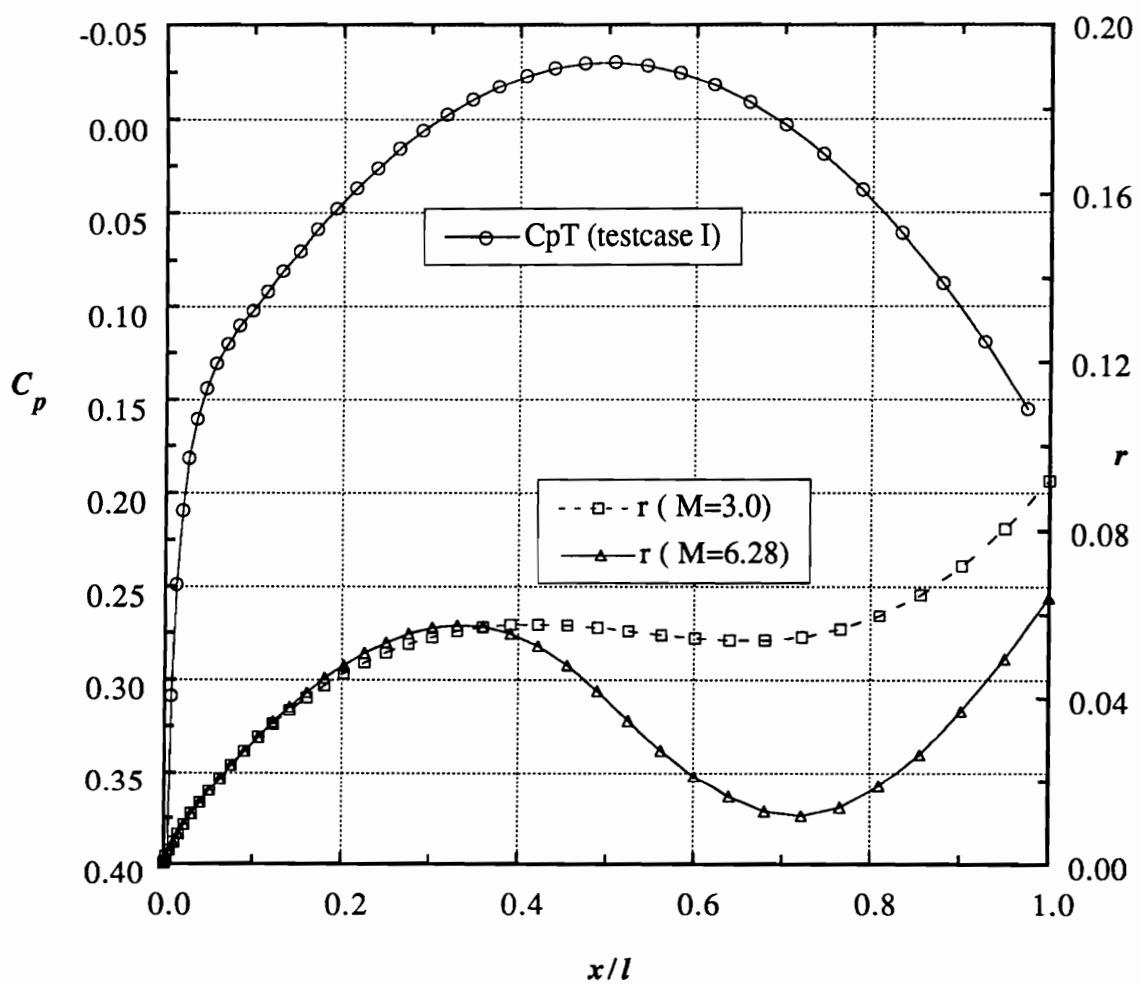


Figure 29. Test Target Pressure I and Resulting Body Shape

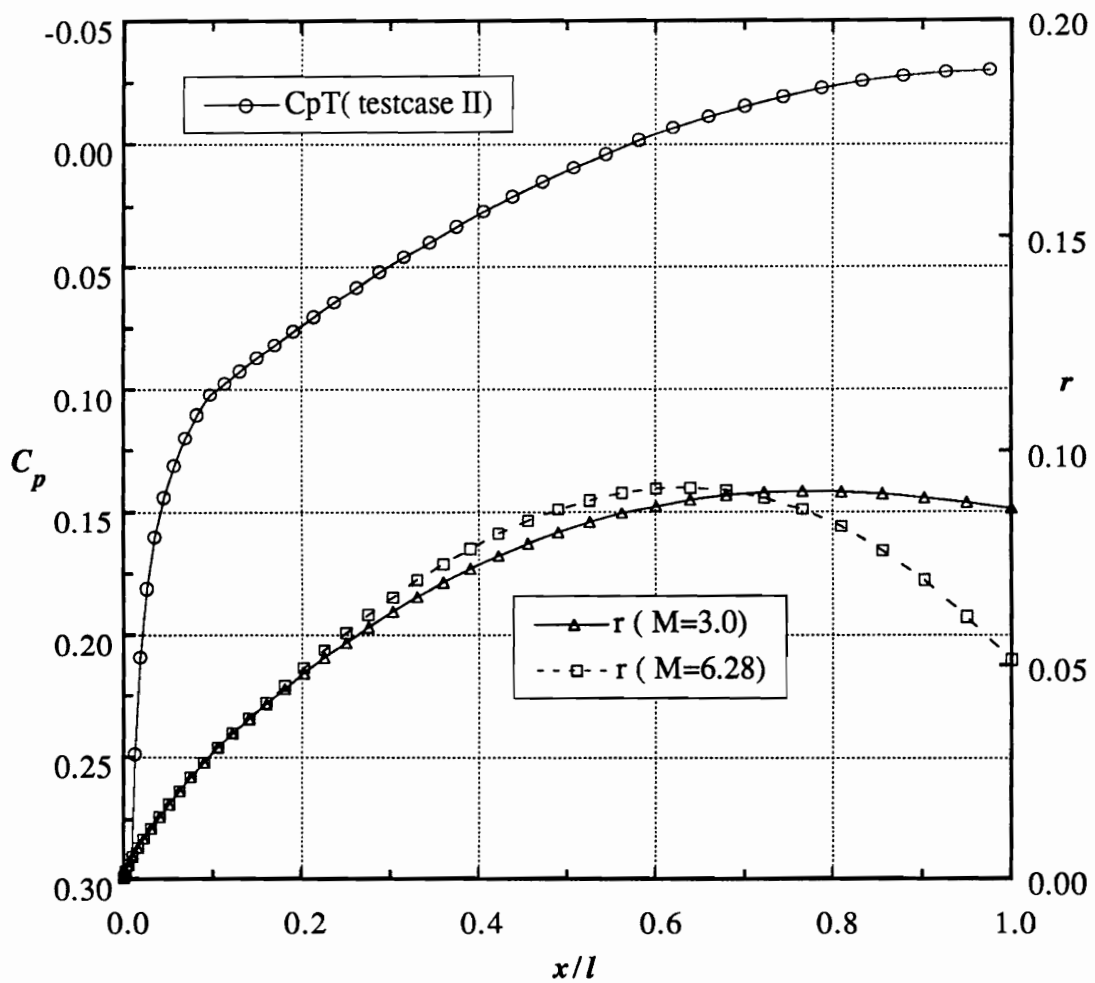


Figure 30. Test Target Pressure II and Resulting Body Shape

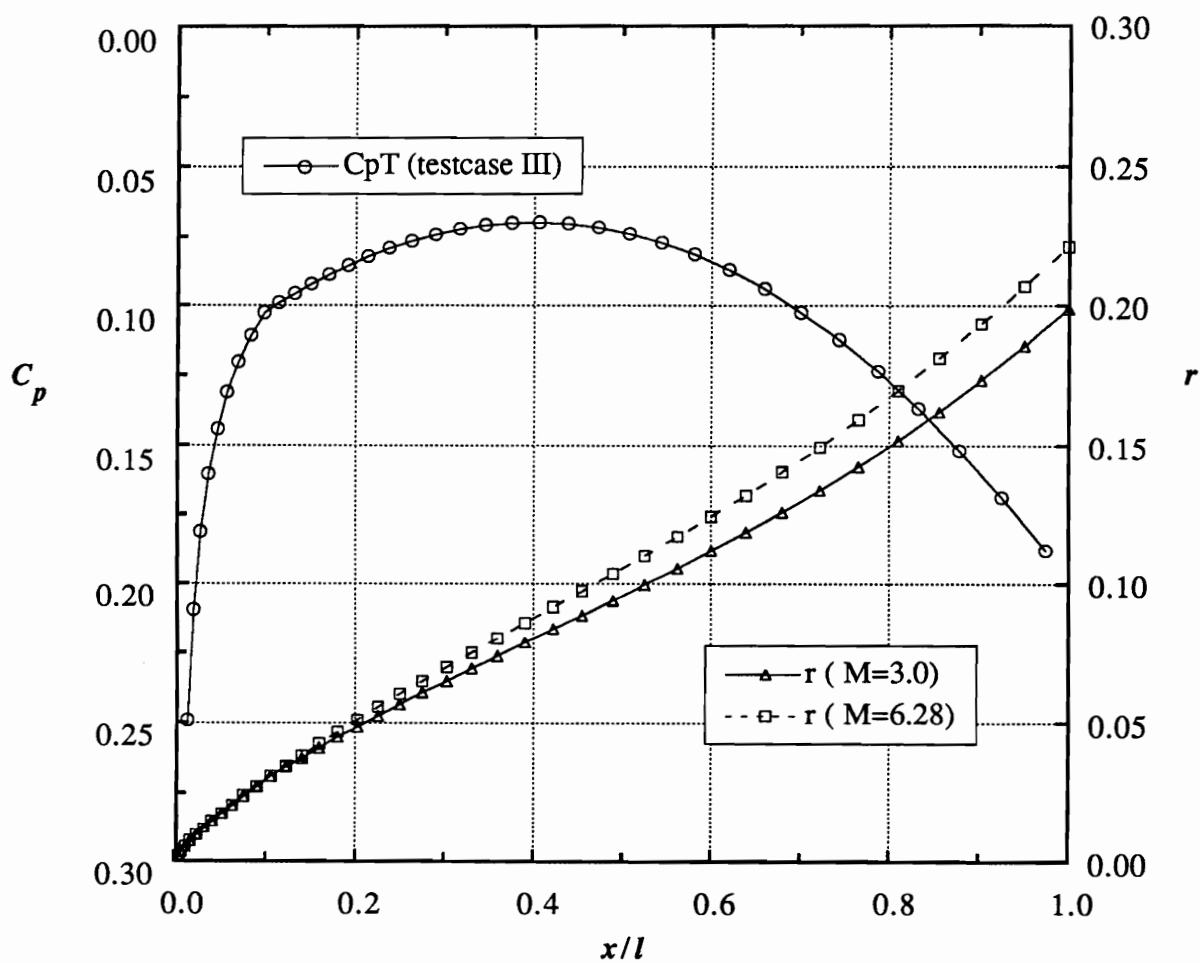


Figure 31. Test Target Pressure III and Resulting Body Shape

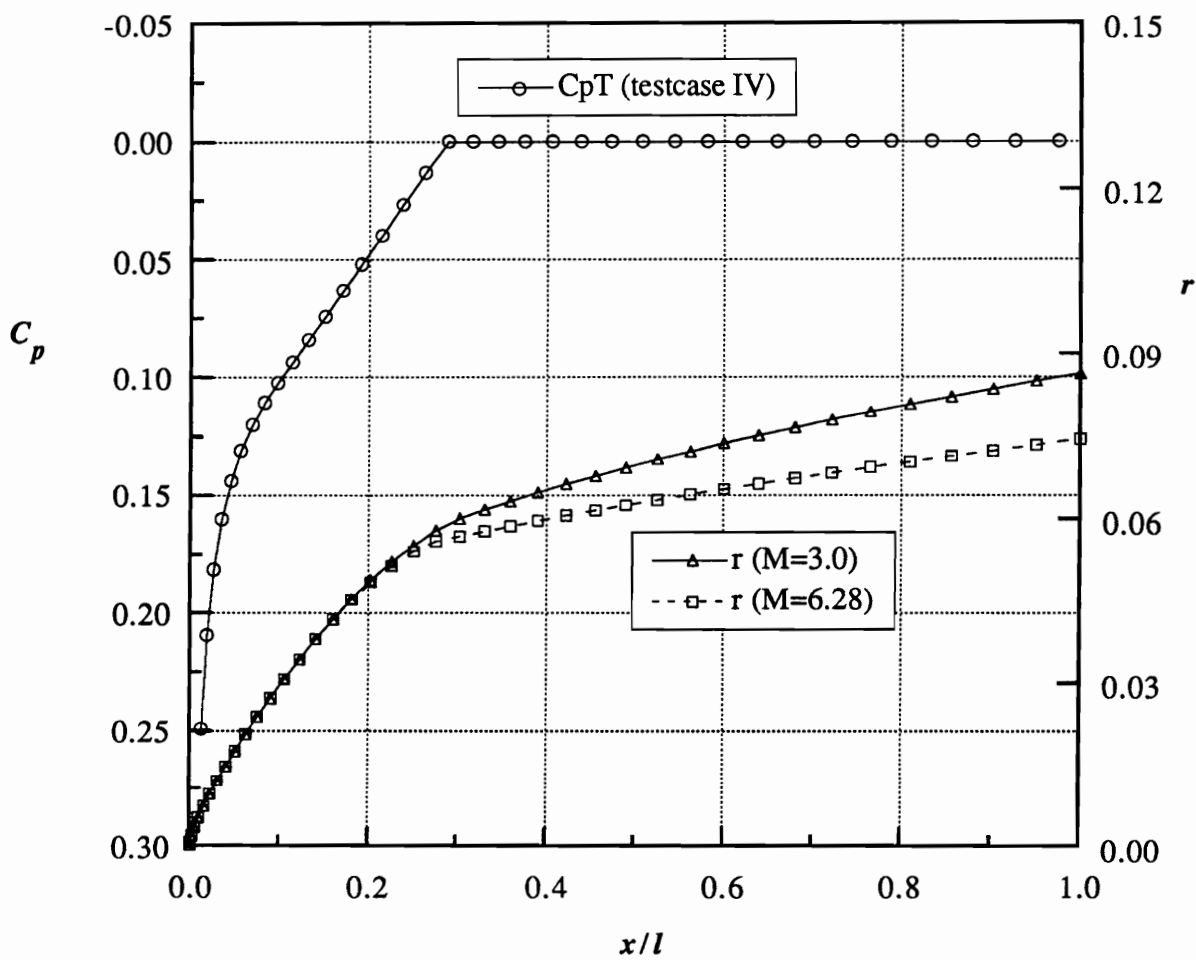


Figure 32. Test Target Pressure IV and Resulting Body Shape

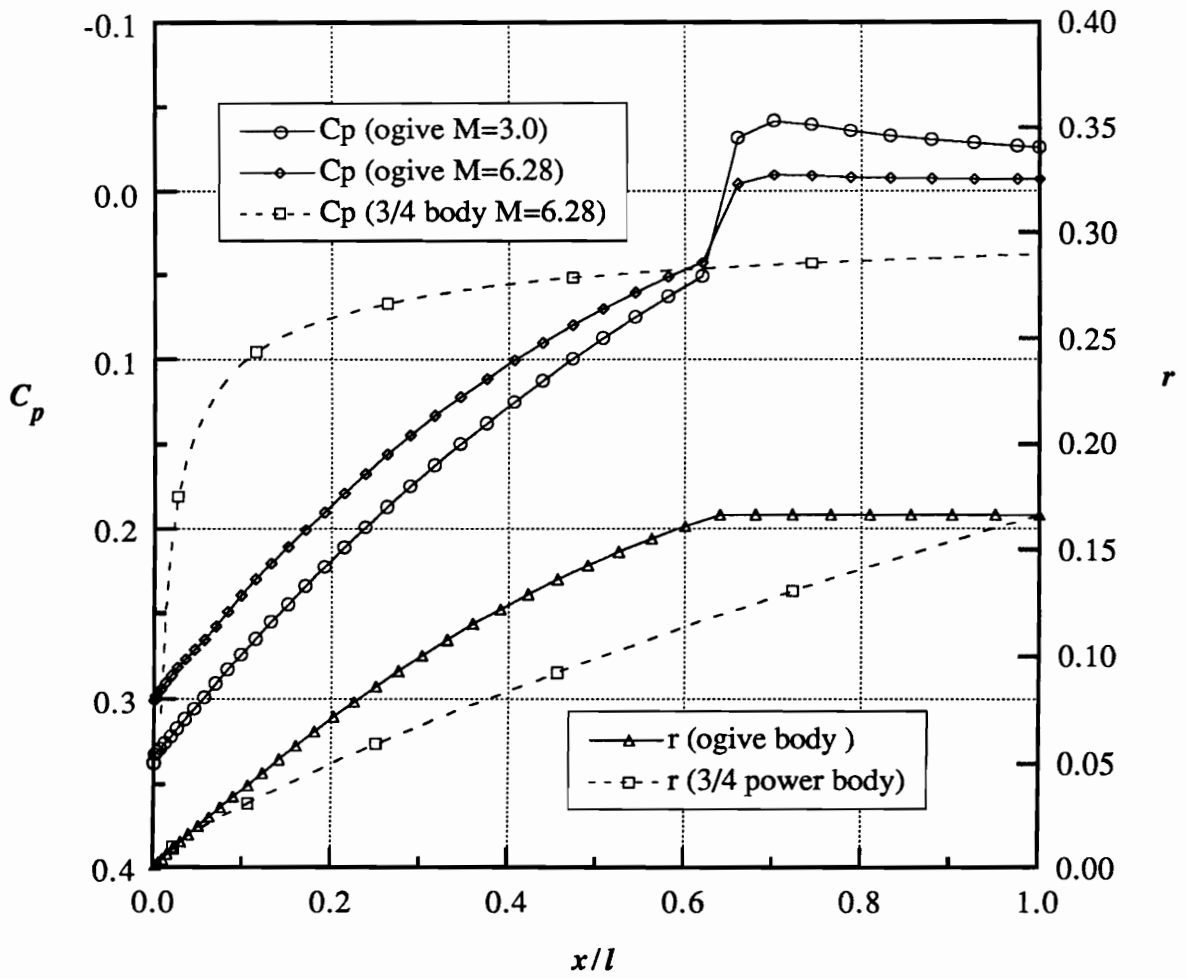


Figure 33. Pressure and Body Geometry of Secant Ogive Cylinder ($l/d = 3.0$)

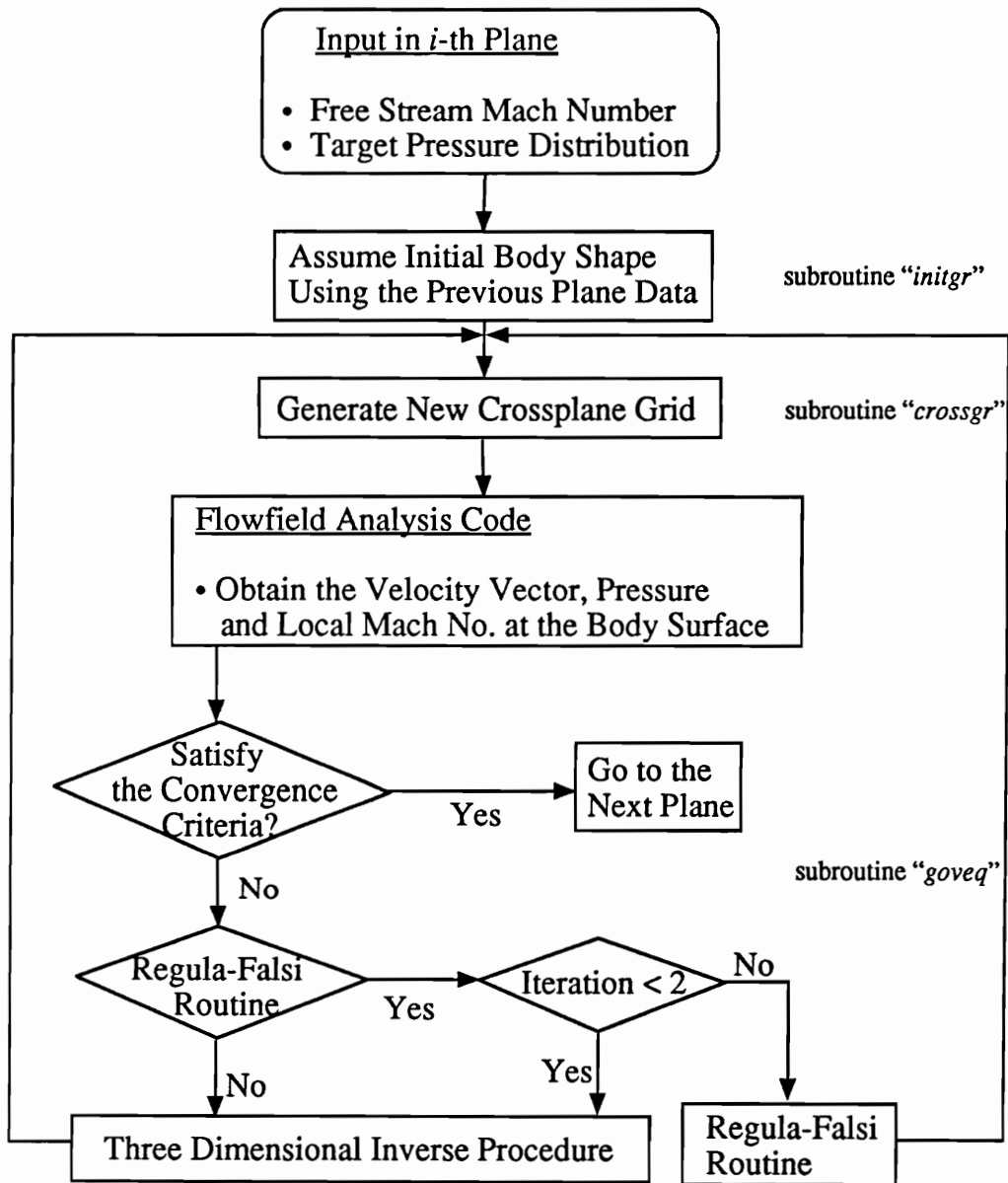


Figure 34. Flow Chart of Three Dimensional Inverse Procedure

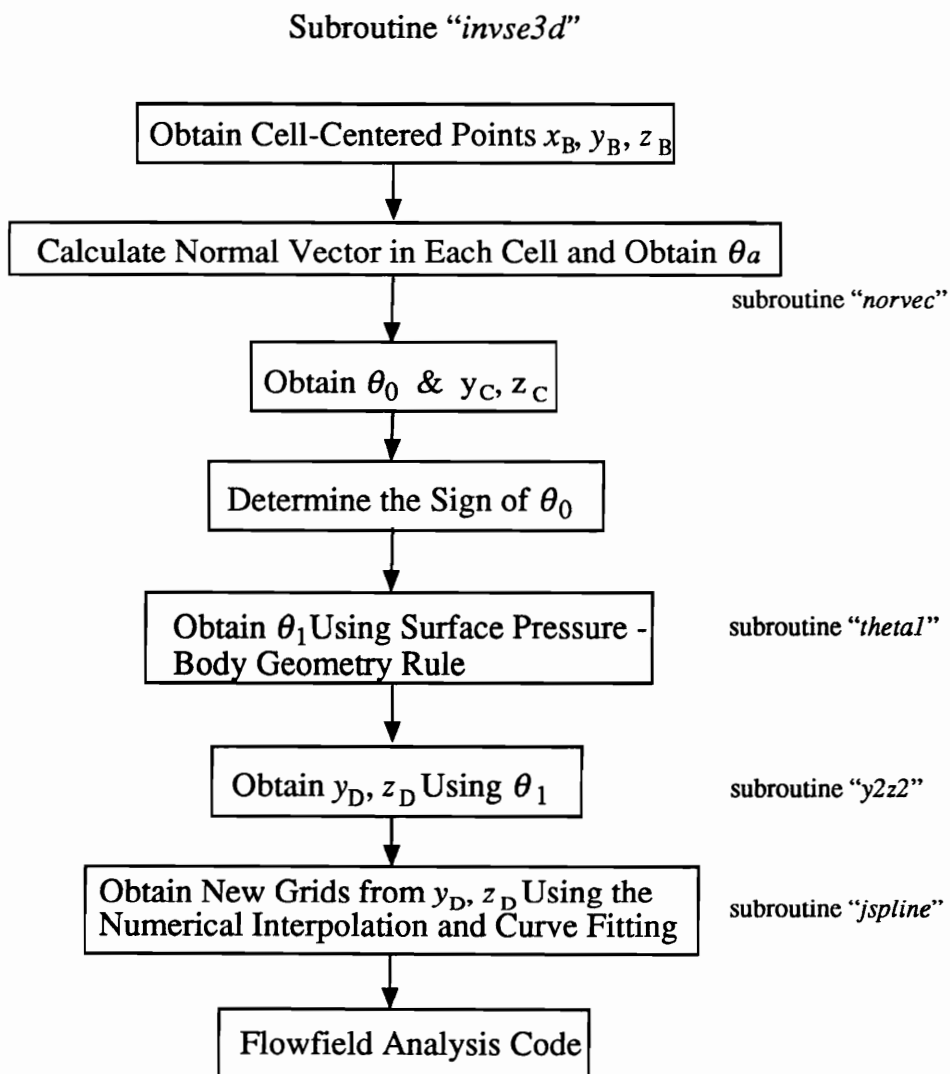
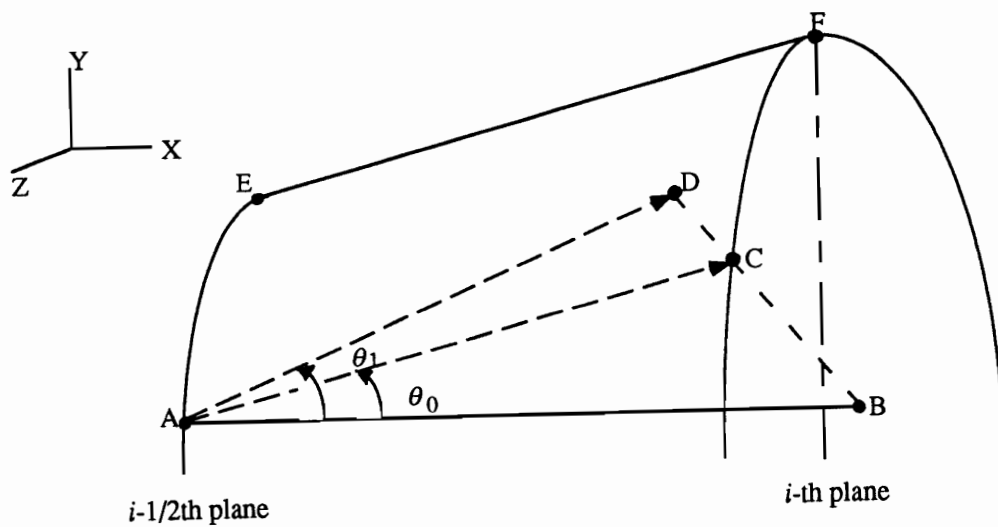
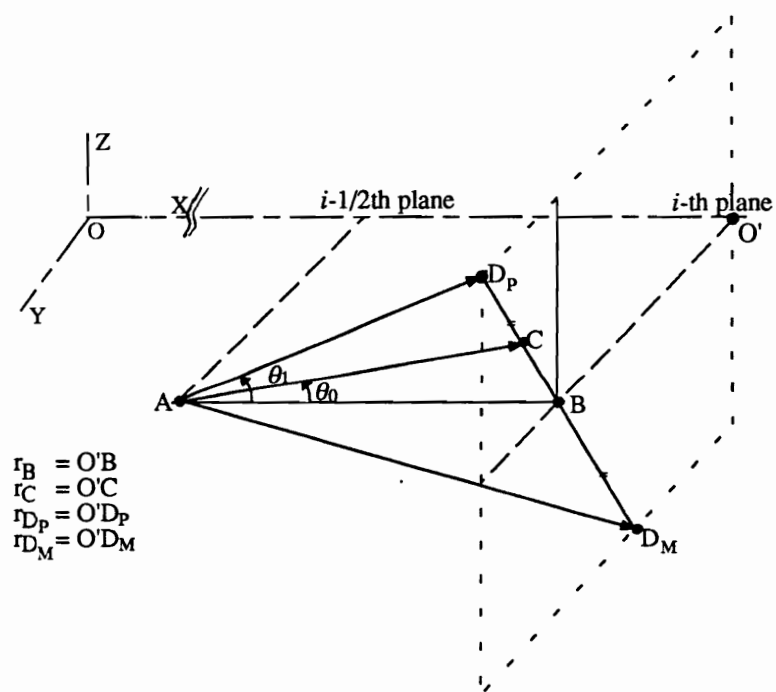


Figure 35. Three Dimensional Inverse Procedure in Detail



(a)



(b)

Figure 36. Body Geometry Correction along the Surface Streamline

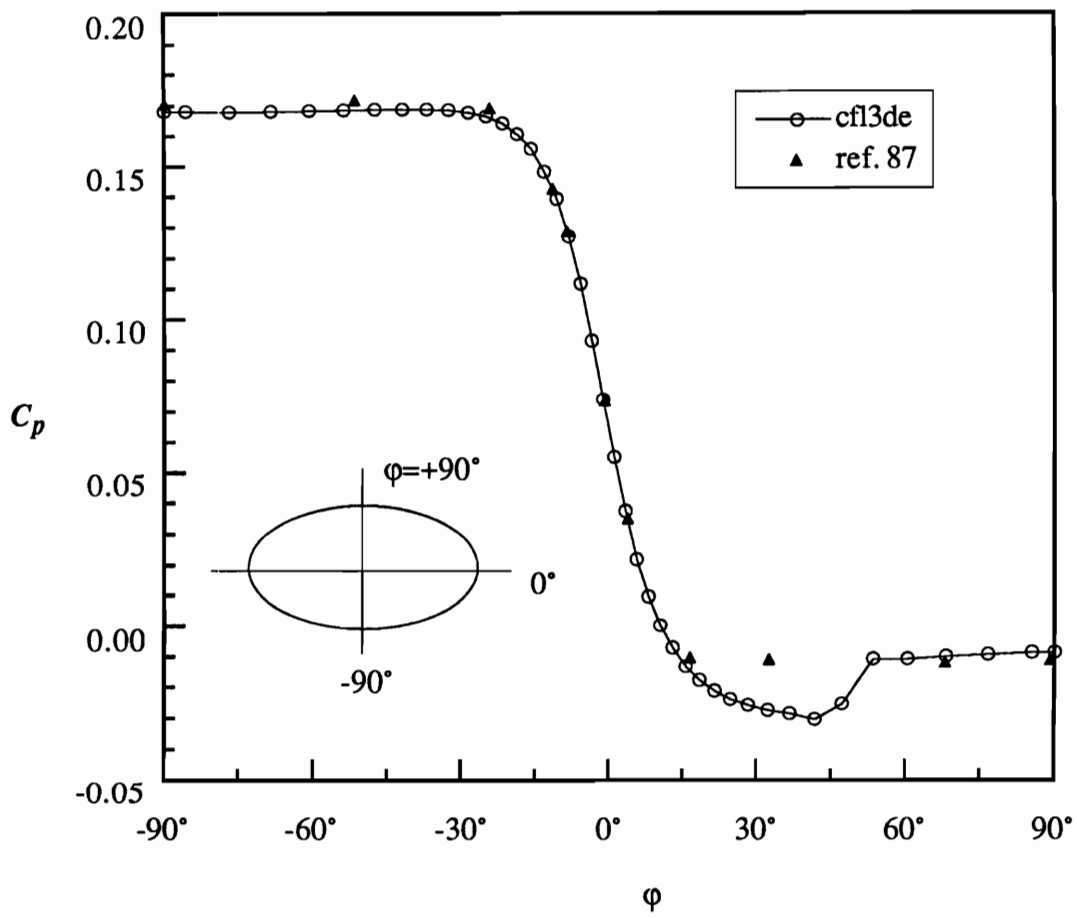


Figure 37. Comparison with Experimental Result for the Elliptic Cone at $M_\infty=5.8$, $\alpha=10^\circ$

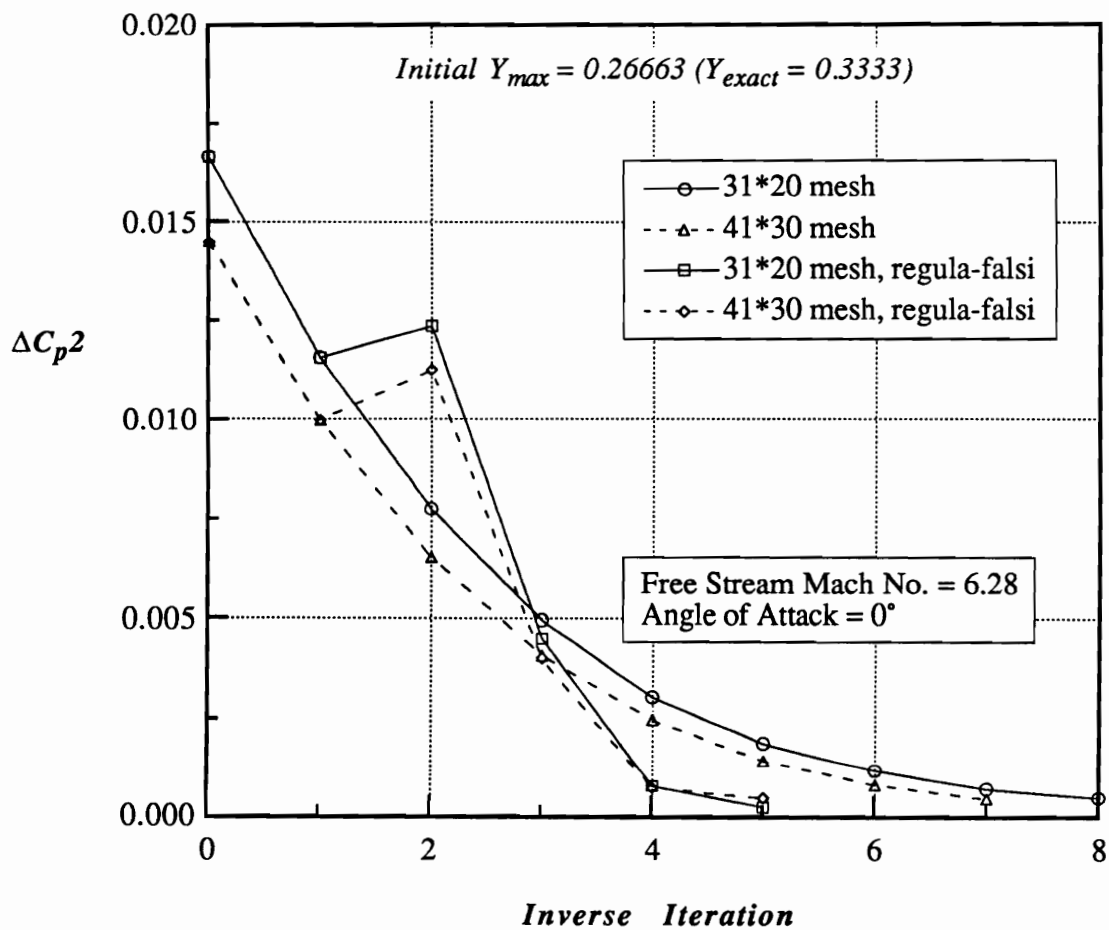


Figure 38. Convergence Behavior of the Elliptic Cone Test Case for Different Grid Densities and Regula-Falsi Option

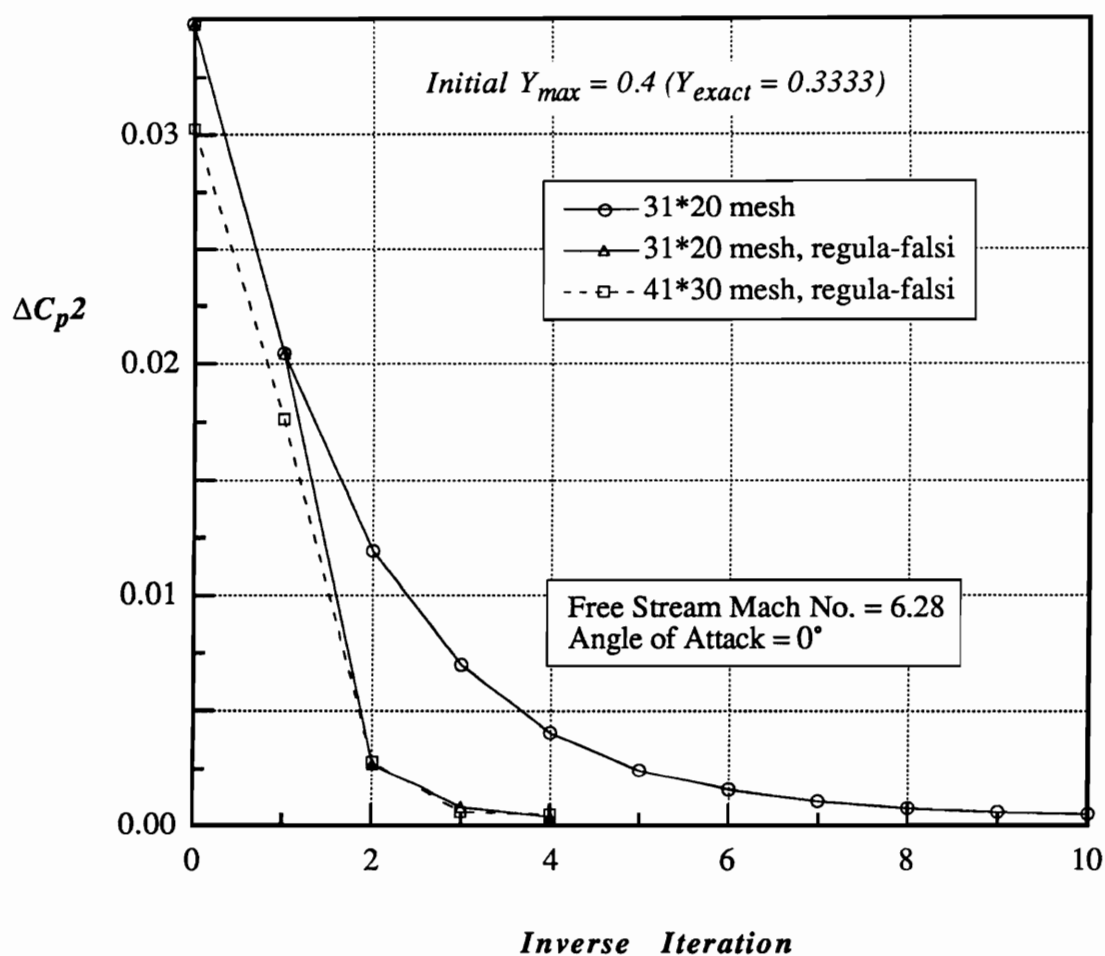


Figure 39. Convergence Behavior of the Elliptic Cone Test Case for Different Grid Densities and Regula-Falsi Option

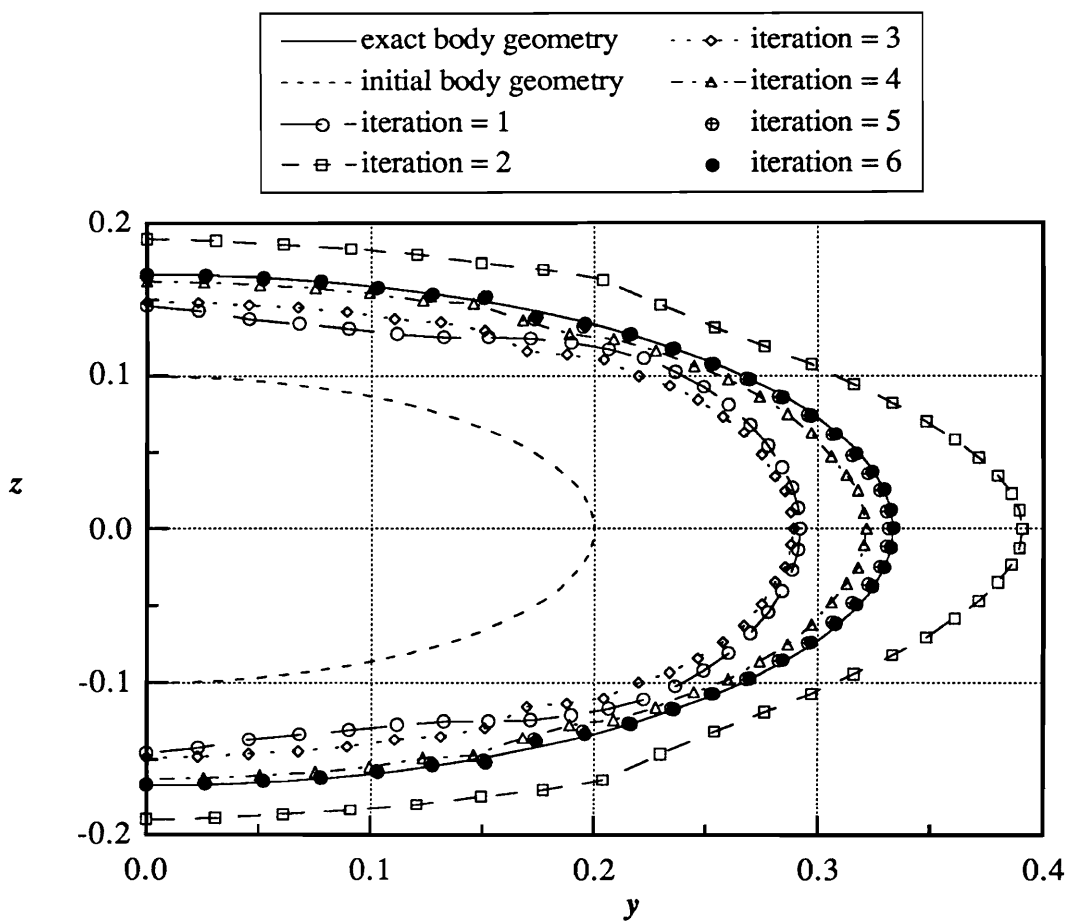


Figure 40. Body Geometry Convergence of the Elliptic Cone Test Case ($M_\infty=6.28$, $\alpha=0^\circ$)

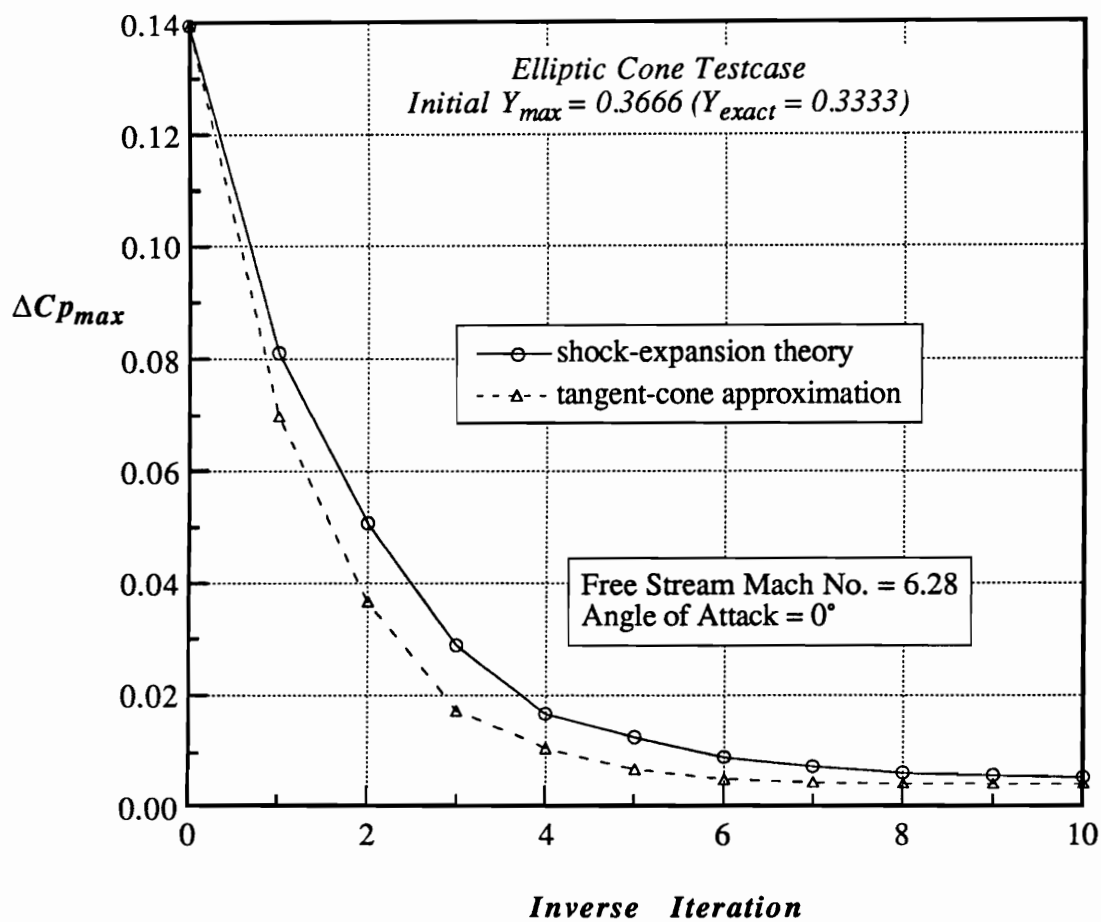


Figure 41. Comparison of Convergence Behavior for
 Two Surface Pressure-Body Geometry Rules

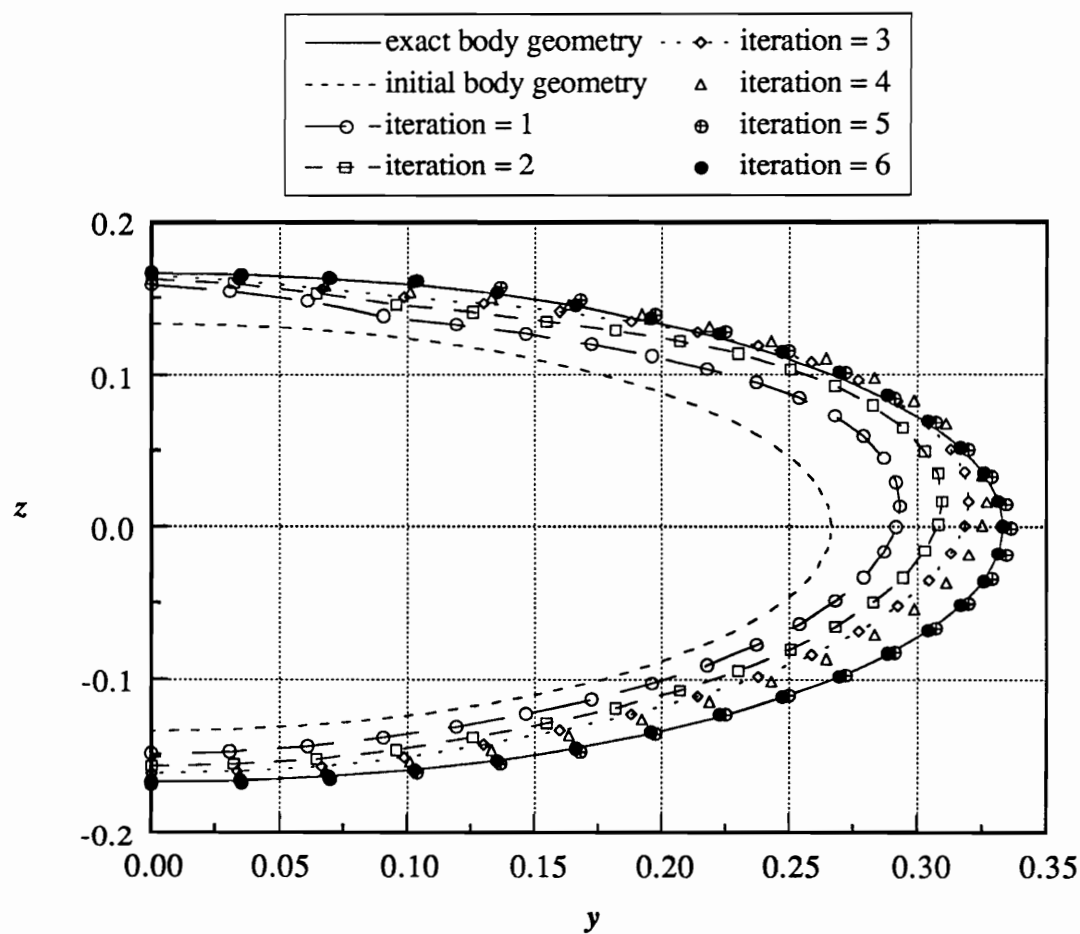


Figure 42. Body Geometry Convergence of the Elliptic Cone Test Case
($M_{\infty}=6.28$, $\alpha=10^{\circ}$)

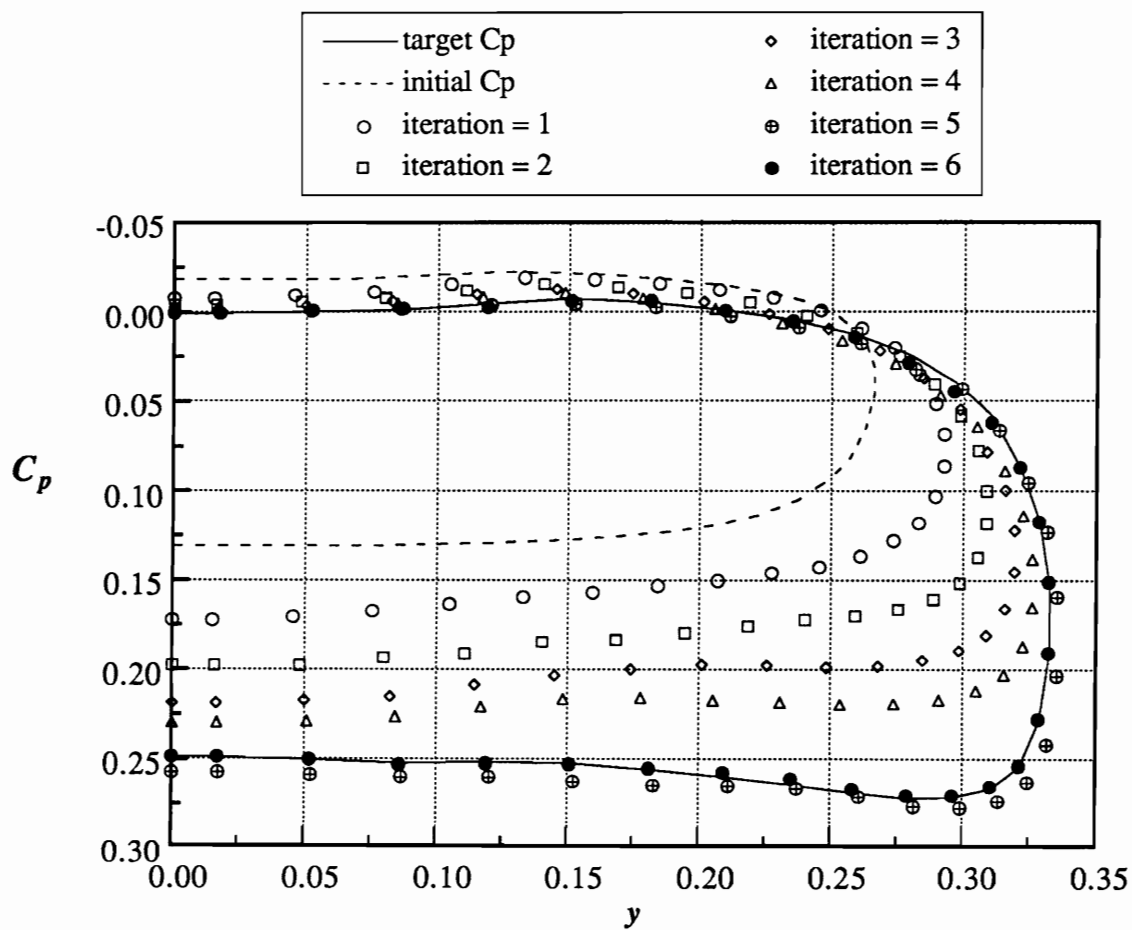


Figure 43. Pressure Coefficient Convergence of the Elliptic Cone Test Case
($M_\infty = 6.28$, $\alpha = 10^\circ$)

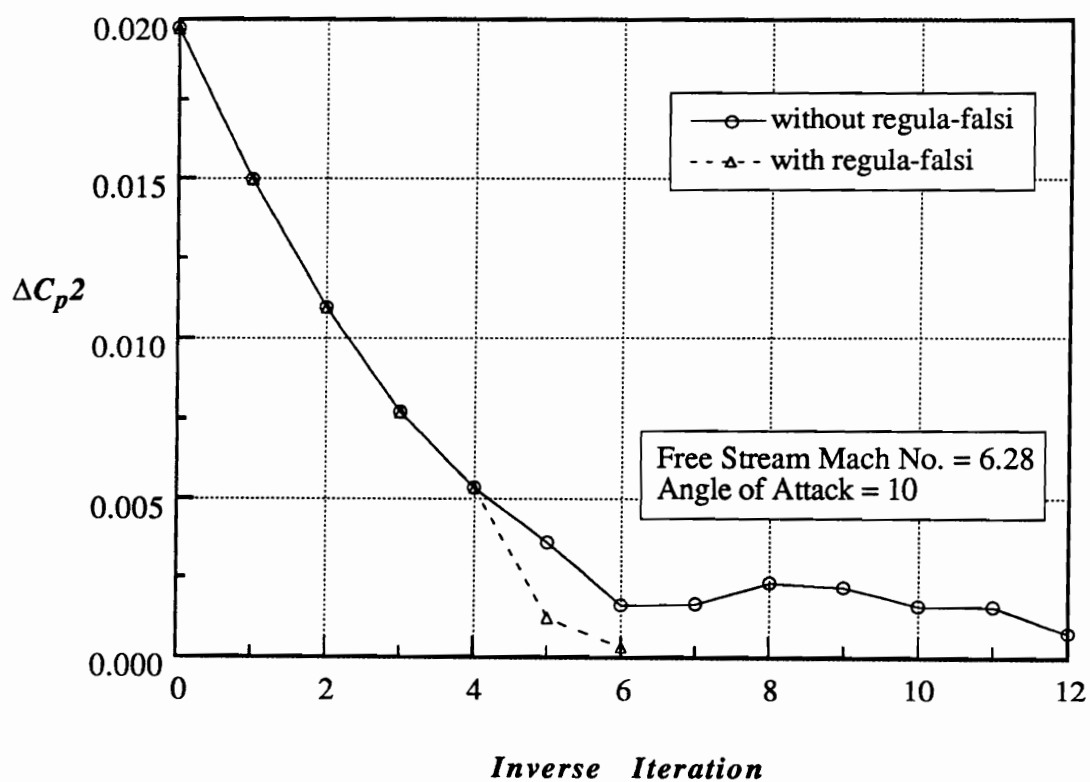


Figure 44. Convergence Behavior of the Elliptic Cone Test Case
with and without Regula- Falsi ($M_{\infty}=6.28$, $\alpha = 10^\circ$)

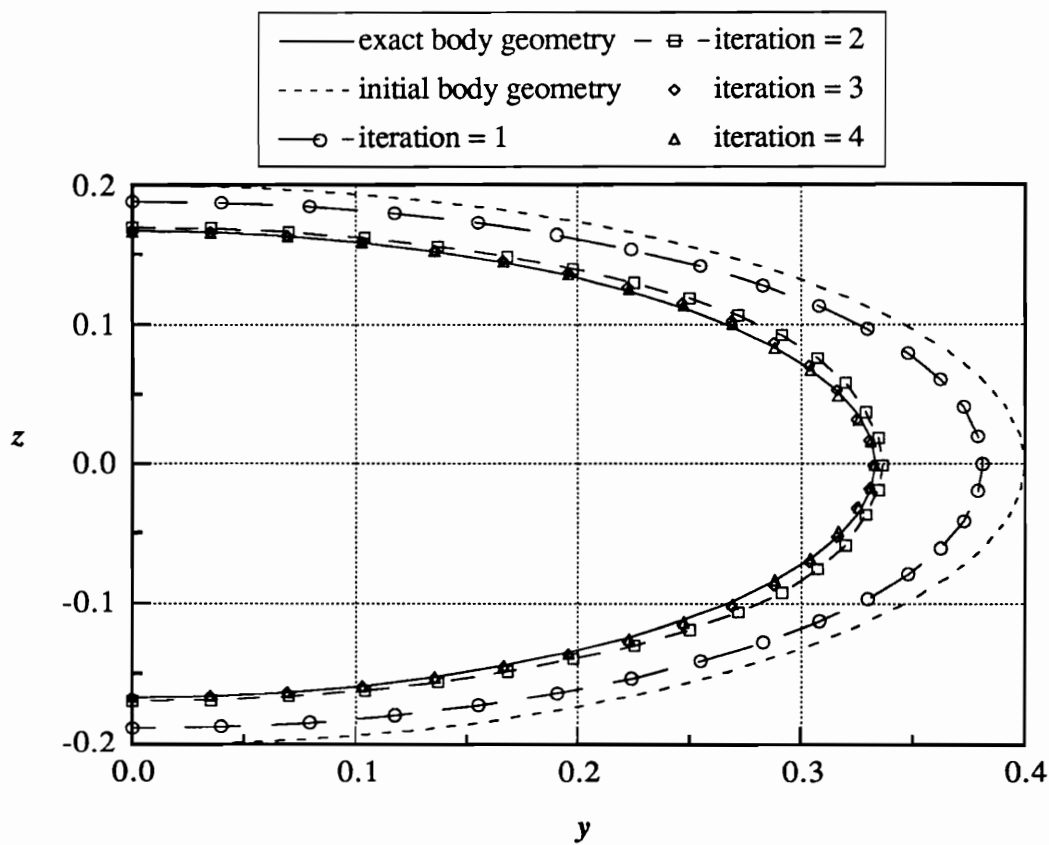


Figure 45. Body Geometry Convergence of the Elliptic Cone Test Case ($M_{\infty}=3.0$, $\alpha=0^\circ$)

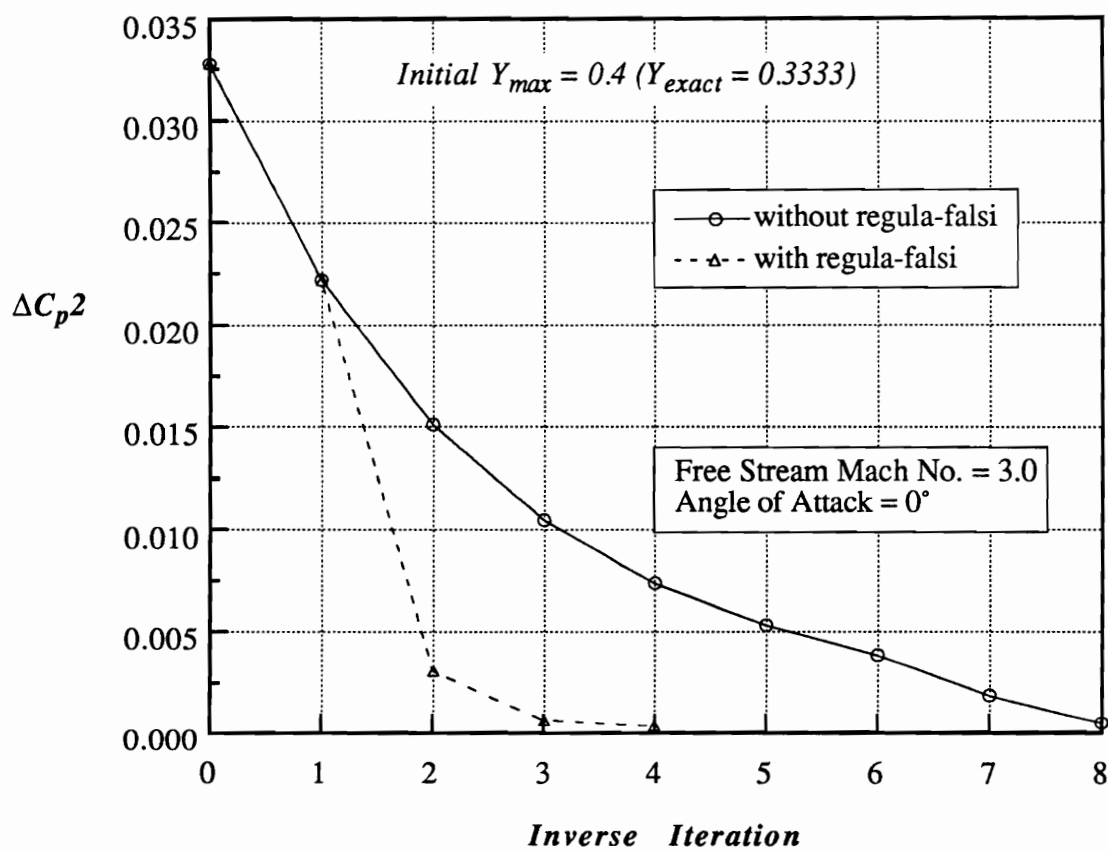


Figure 46. Convergence Behavior of the Elliptic Cone Test Case
with and without Regula-Falsi

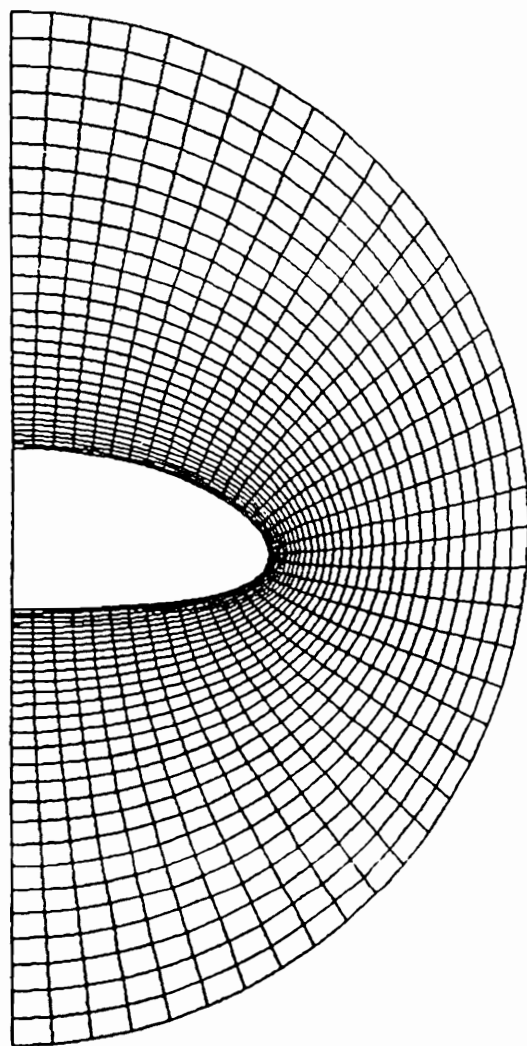


Figure 47. Cross-Flow Plane Computational Grids for
the 'Wing Like' Nonconical Cambered Body at $x/l = 1.0$

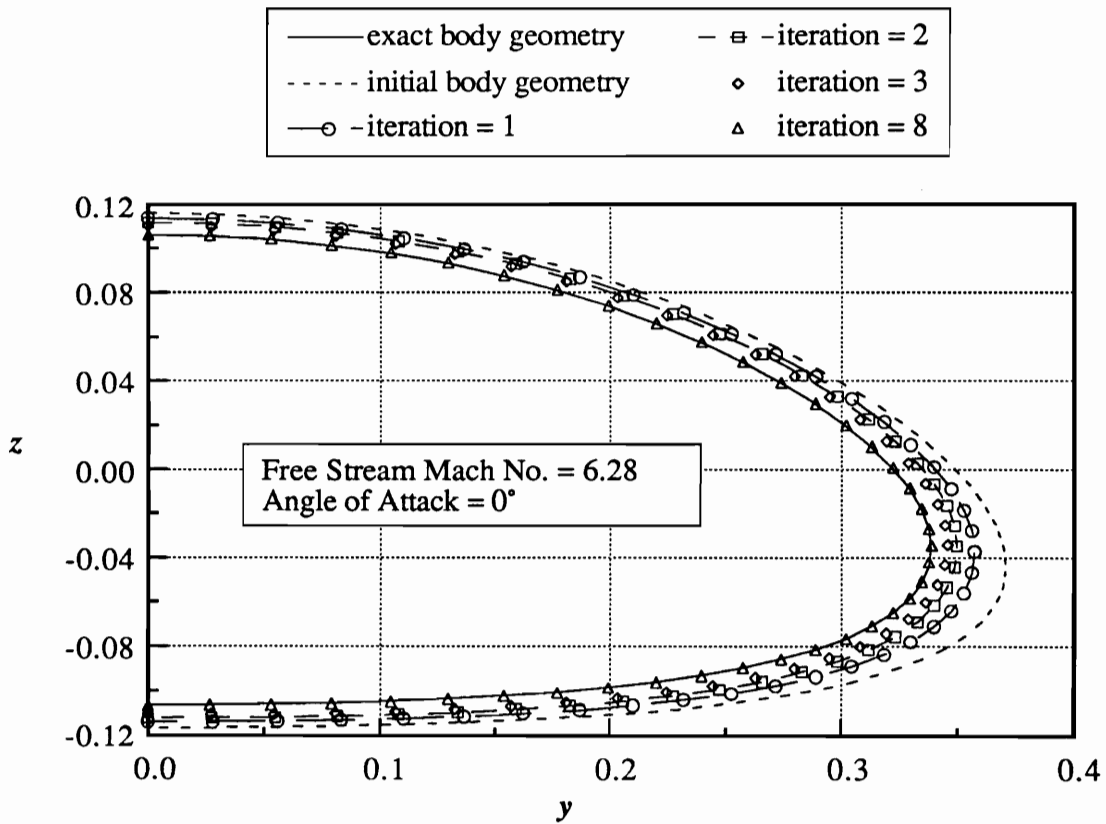


Figure 48. Body Geometry Convergence of the 'Wing Like' Nonconical Cambered Body

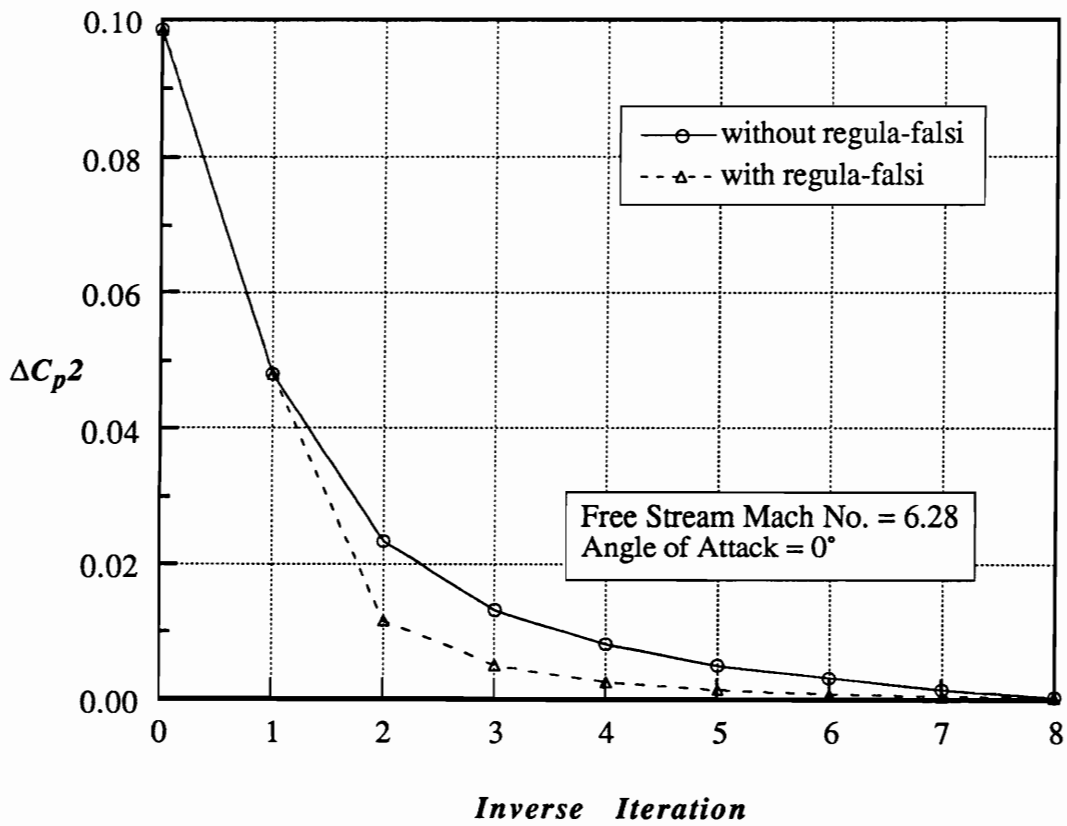


Figure 49. Convergence Behavior of the 'Wing Like' Nonconical Cambered Body with and without Regula-Falsi Scheme

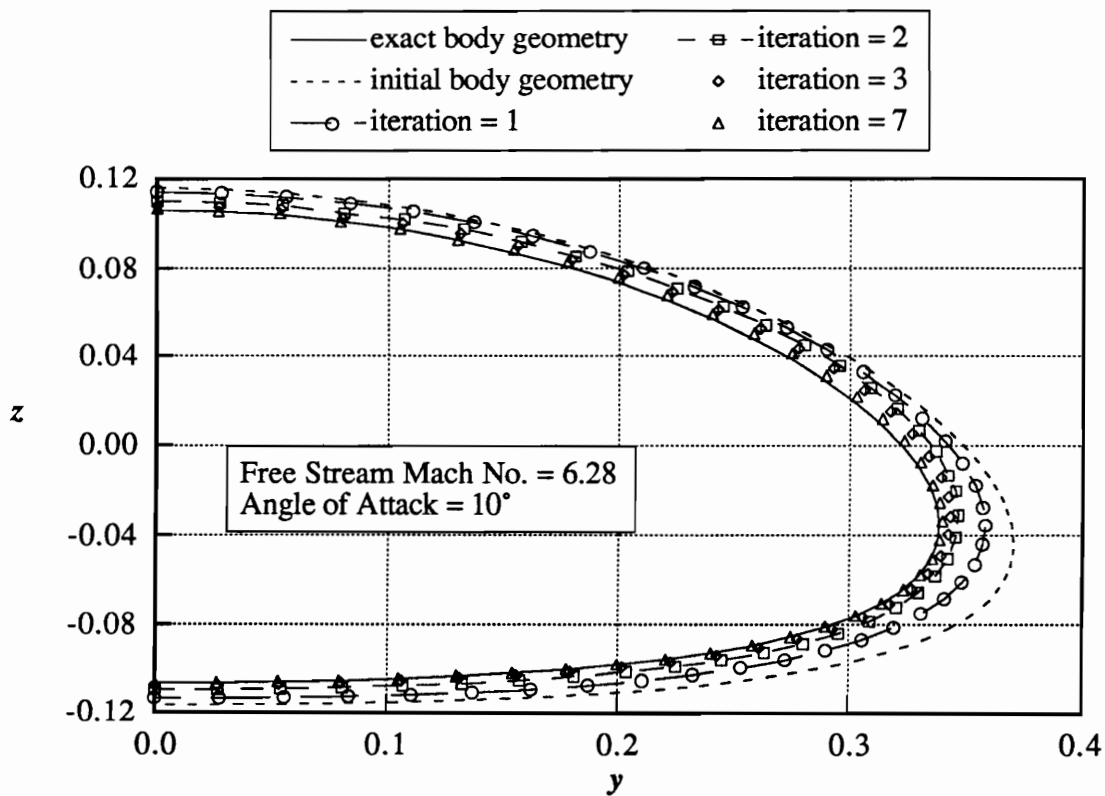
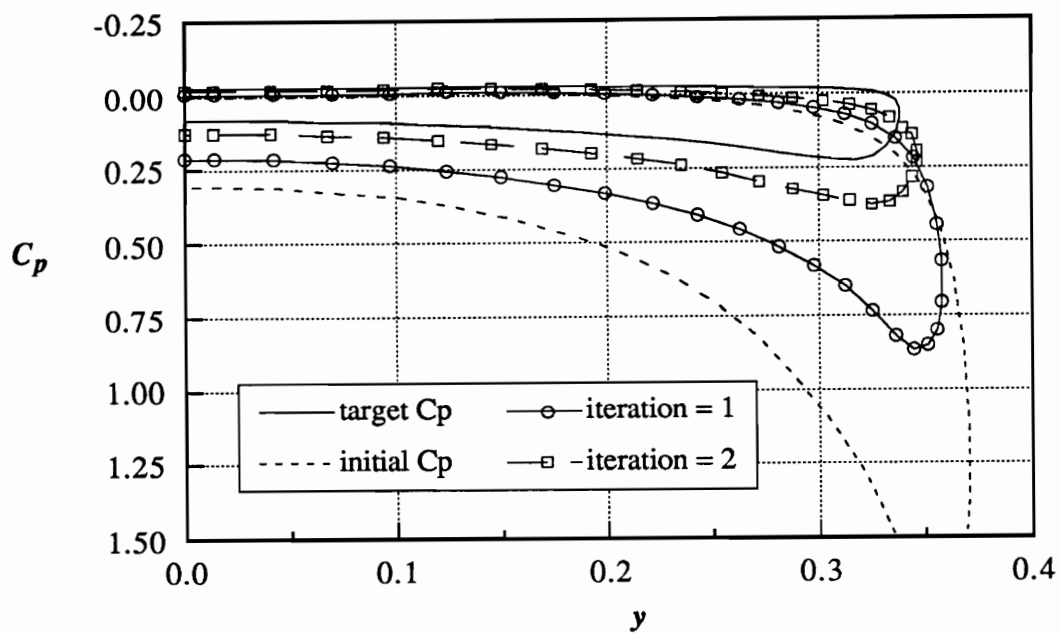
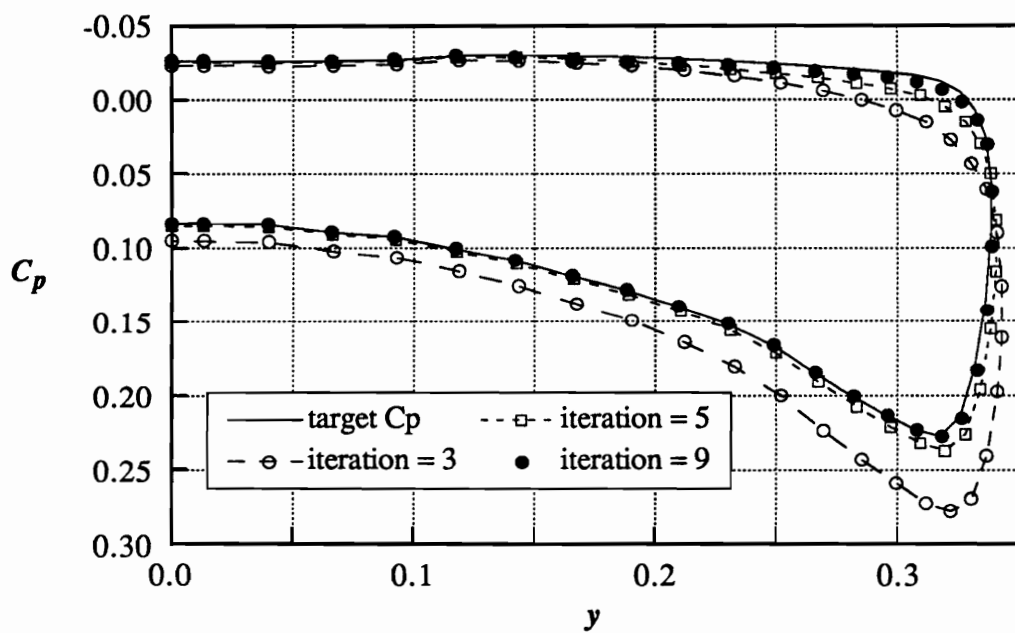


Figure 50. Body Geometry Convergence of the 'Wing Like' Nonconical Cambered Body



(a) Inverse Iteration 0 - 2



(b) Inverse Iteration 3 - 9

Figure 51. Pressure Coefficient Convergence of the 'Wing Like' Nonconical Cambered Body ($M_\infty=6.28$, $\alpha=10^\circ$)

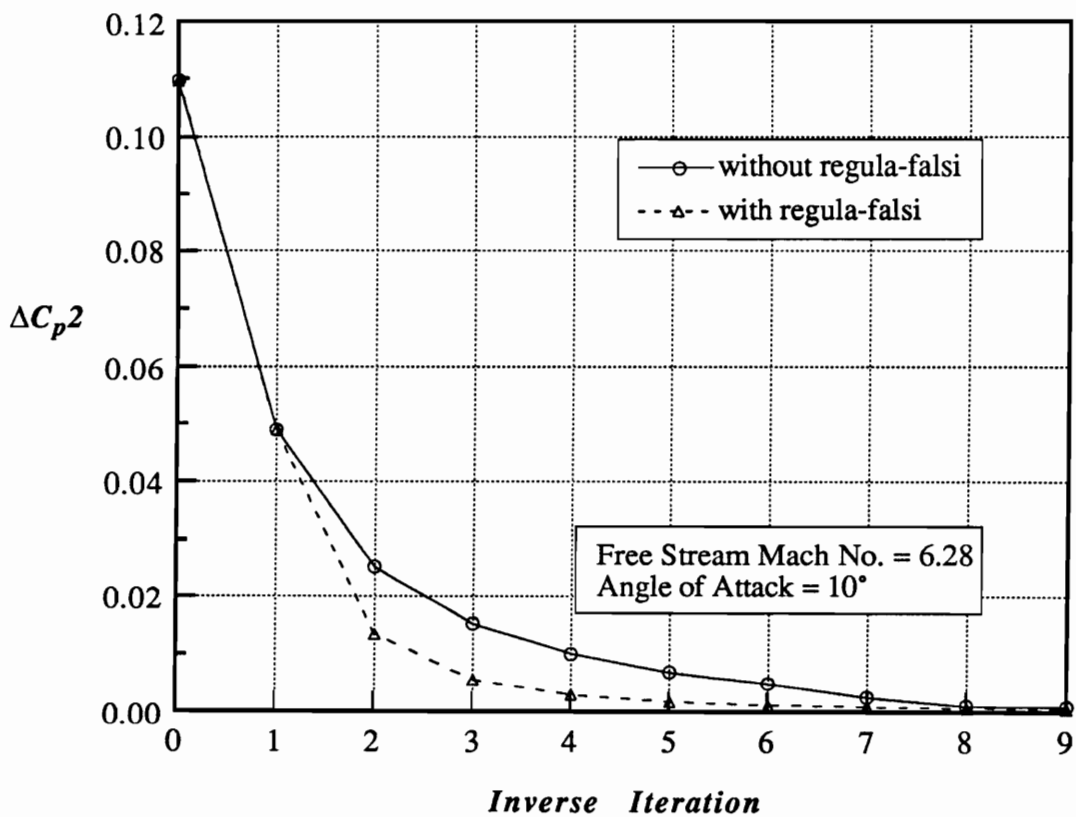


Figure 52. Convergence Behavior of the 'Wing Like' Nonconical Cambered Body with and without Regula-Falsi

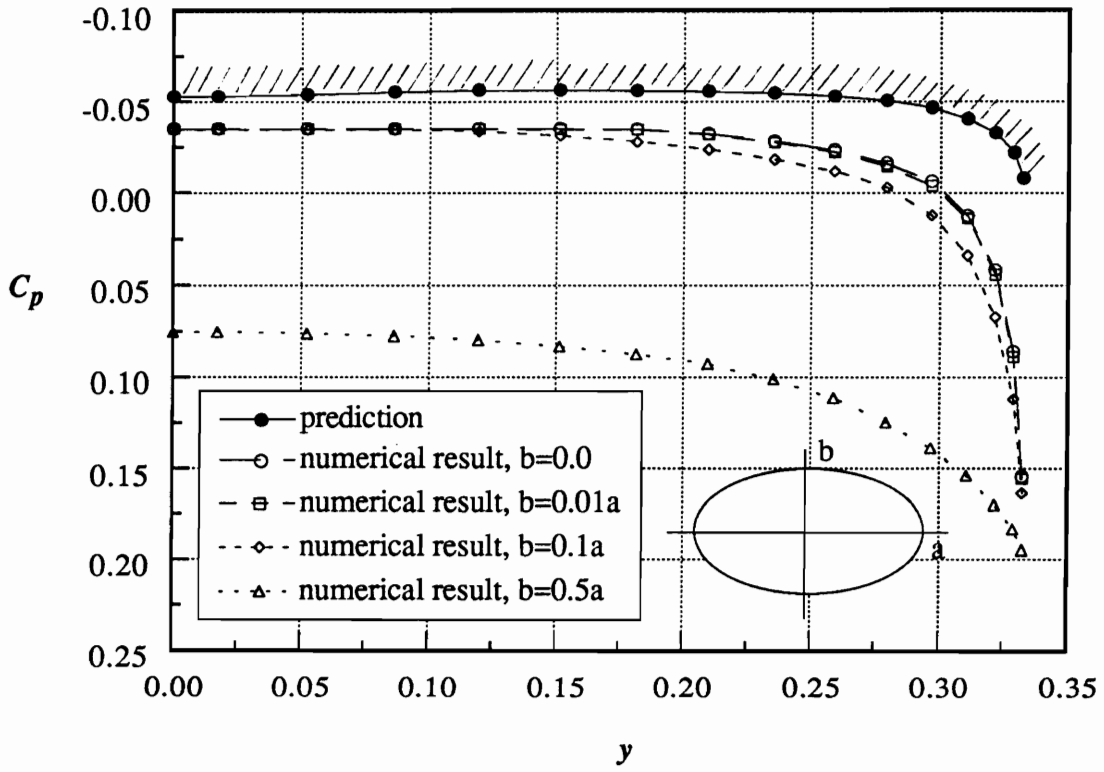
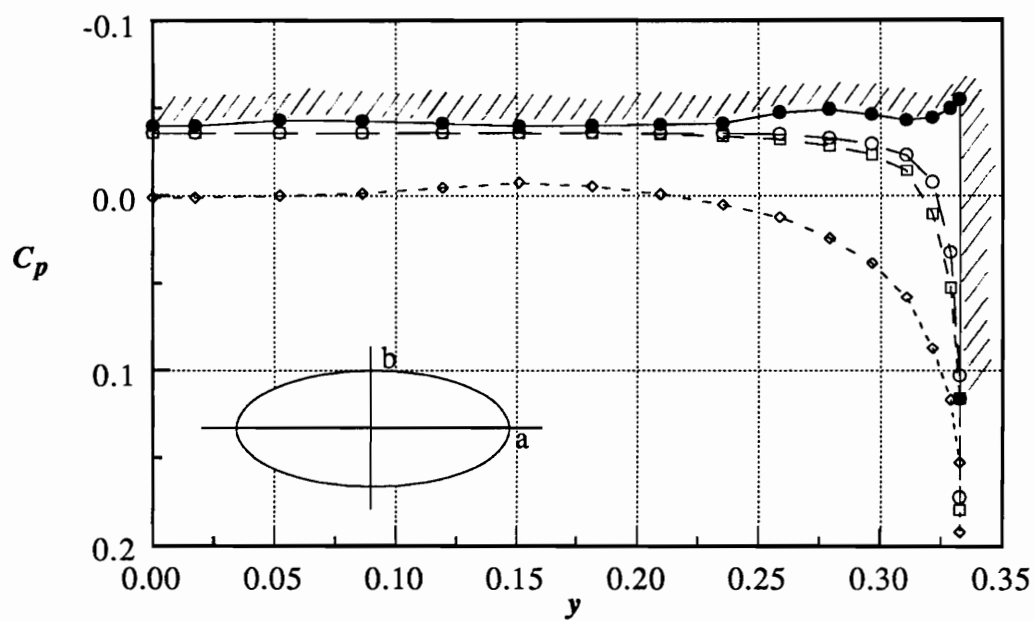
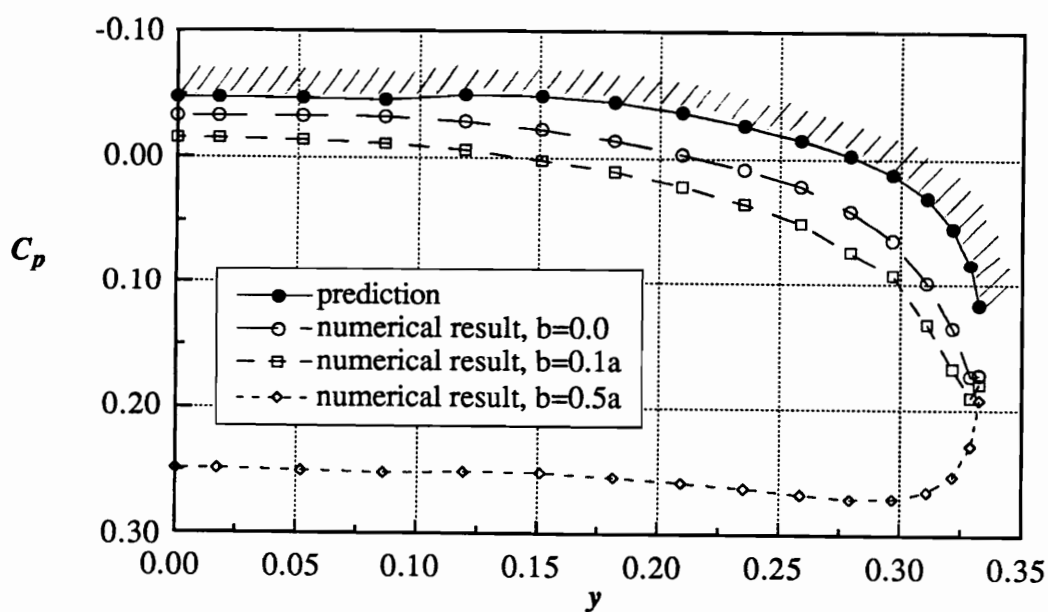


Figure 53. Domain of Existence (Elliptic Cone: $M_\infty=6.28$, $\alpha = 0^\circ$)

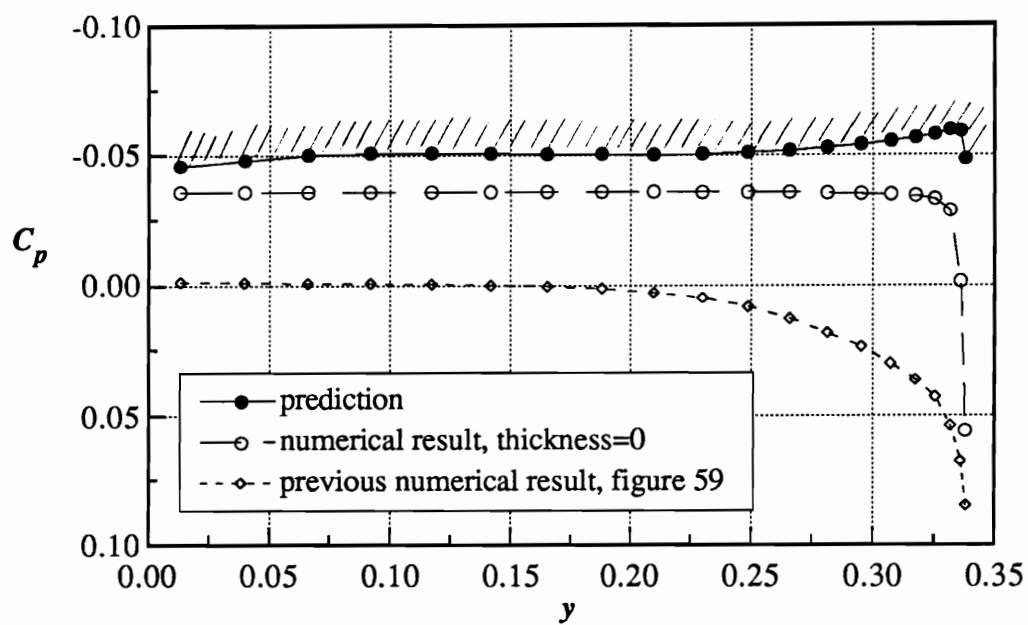


(a) Upper Surface

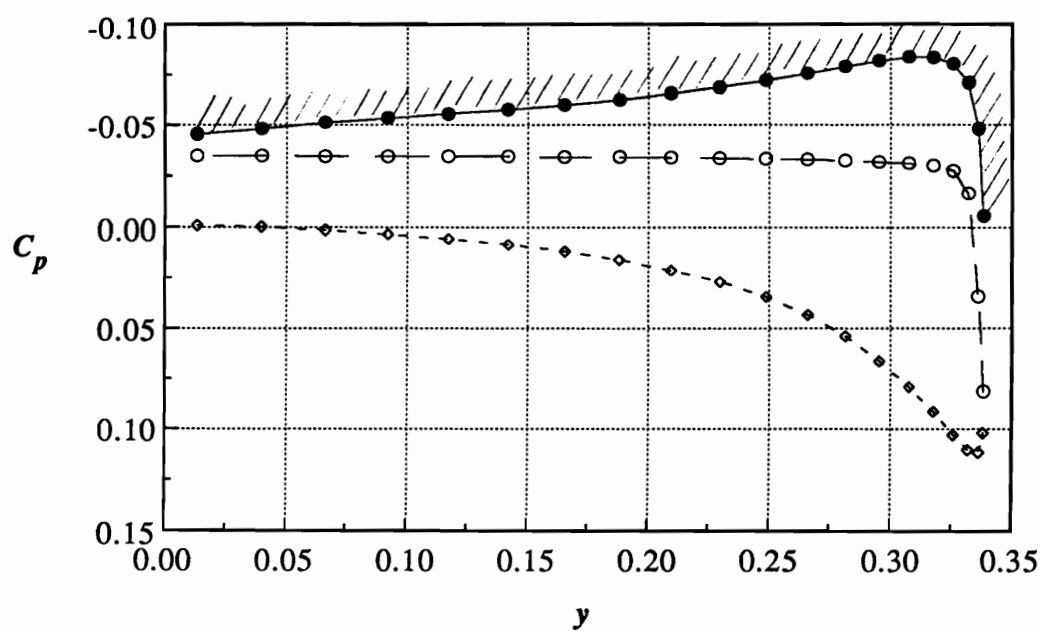


(b) Lower Surface

Figure 54. Domain of Existence (Elliptic Cone: $M_\infty=6.28$, $\alpha = 10^\circ$)

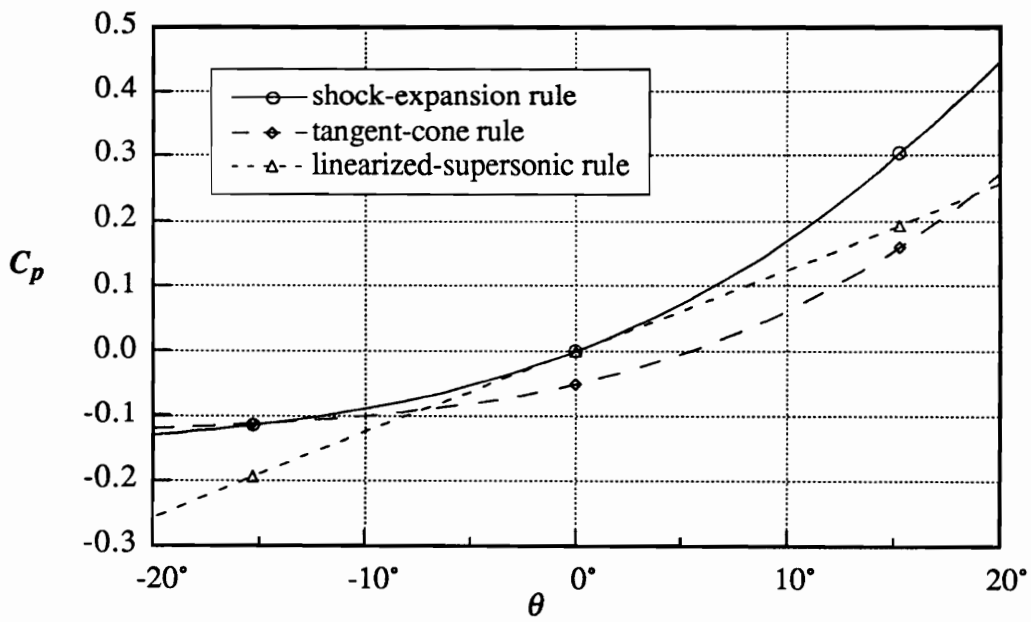


(a) Upper Surface

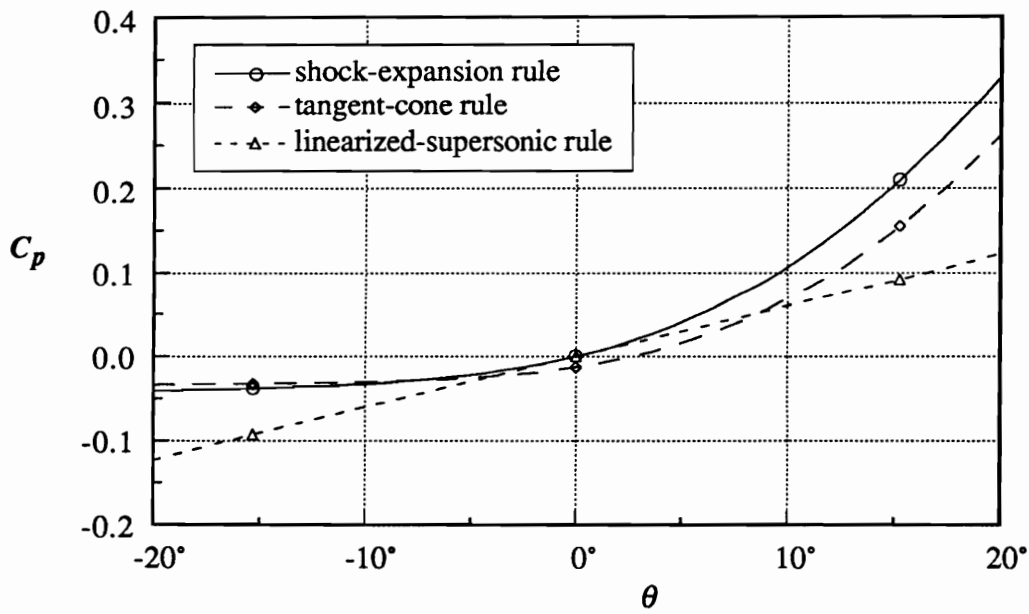


(b) Lower Surface

Figure 55. Domain of Existence (Cambered Body: $M_\infty=6.28$, $\alpha = 0^\circ$)



(a) Mach Number = 3.0



(b) Mach Number = 6.0

Figure 56. Pressure Coefficient as a Function of Surface Slope
for Several Pressure-Shape Rules

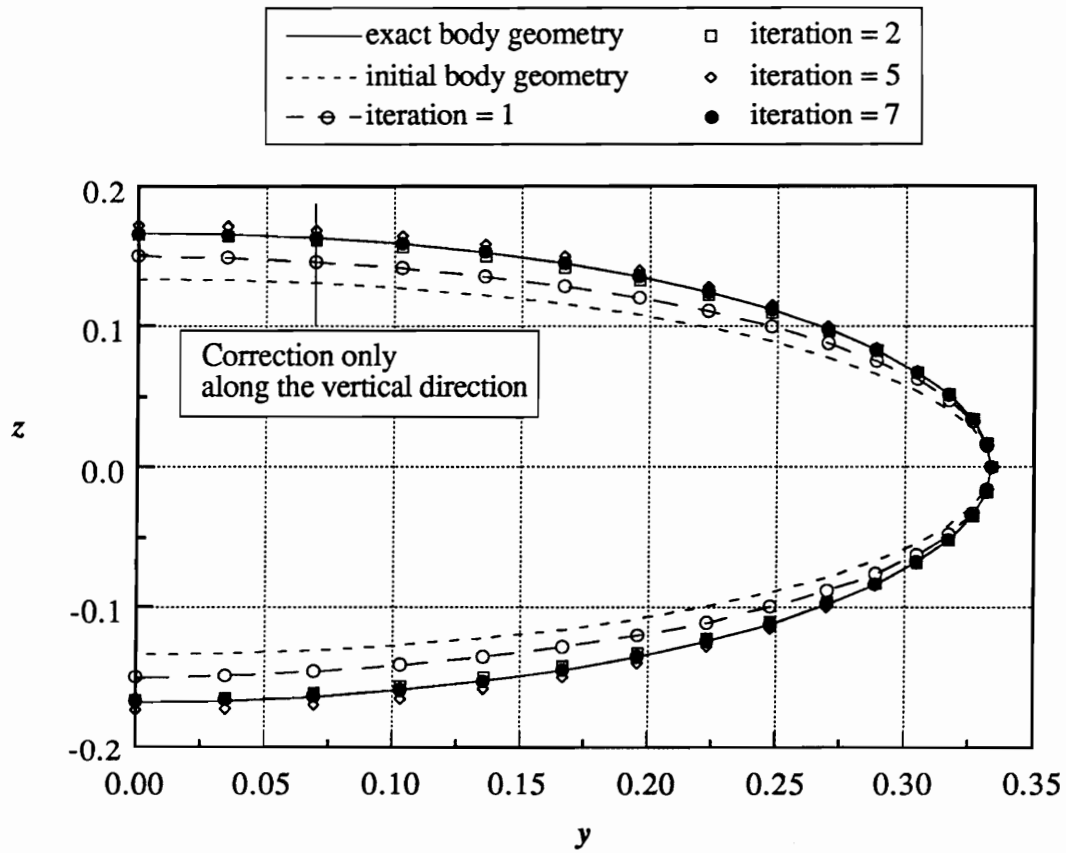


Figure 57. Effect of Fixed Planform:

Body Geometry Convergence of Elliptic Cone ($M_\infty=6.28$, $\alpha = 0^\circ$)

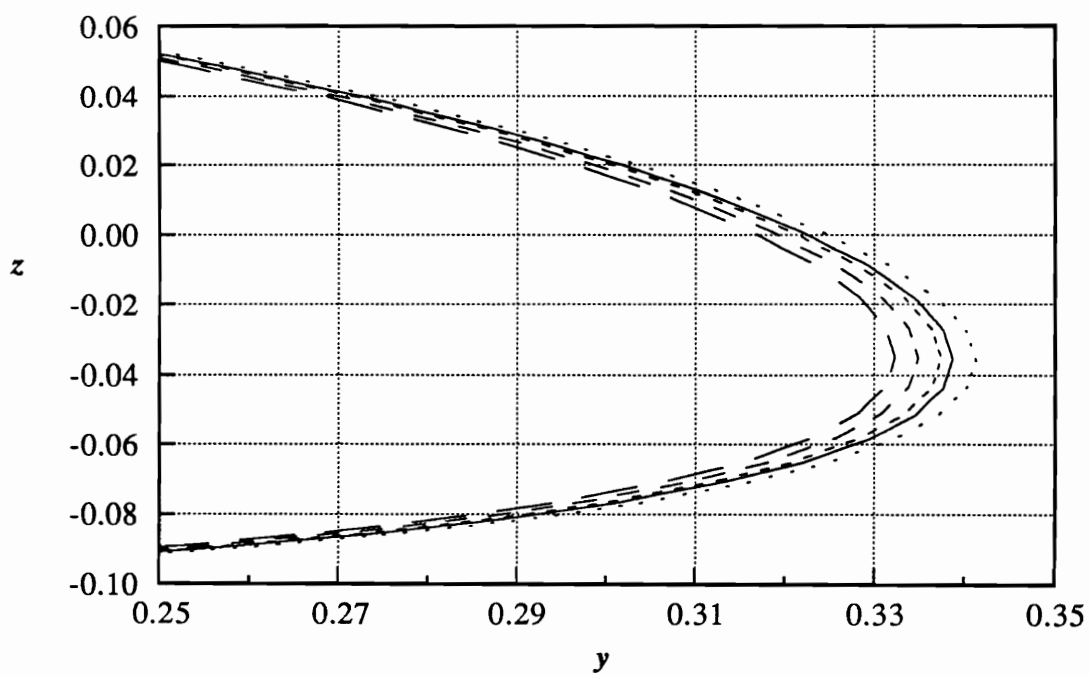
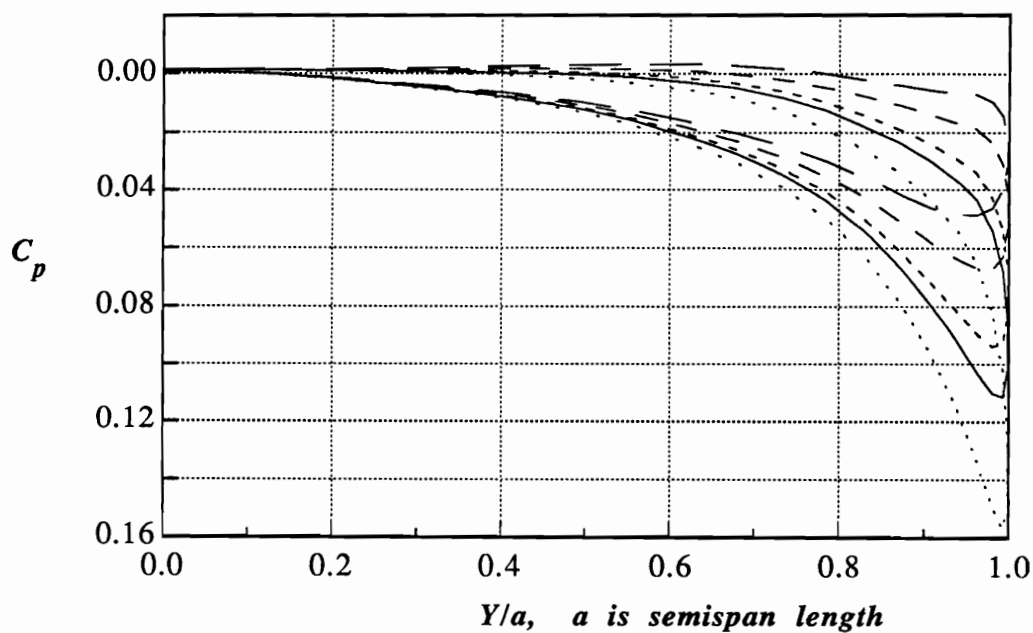
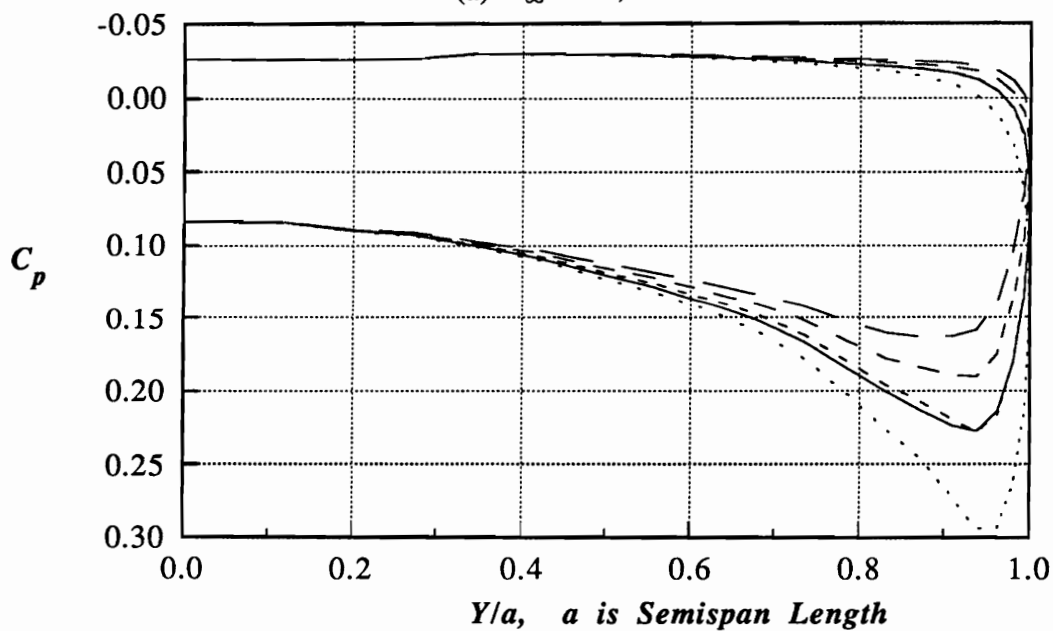


Figure 58. 'Wing Like' Nonconical Cambered Body with Different Semispan Lengths



(a) $M_\infty = 6.28, \alpha = 0^\circ$



(b) $M_\infty = 6.28, \alpha = 10^\circ$

Figure 59. Several C_p Distributions with Different Semispan Lengths in 'Wing Like' Nonconical Cambered Body

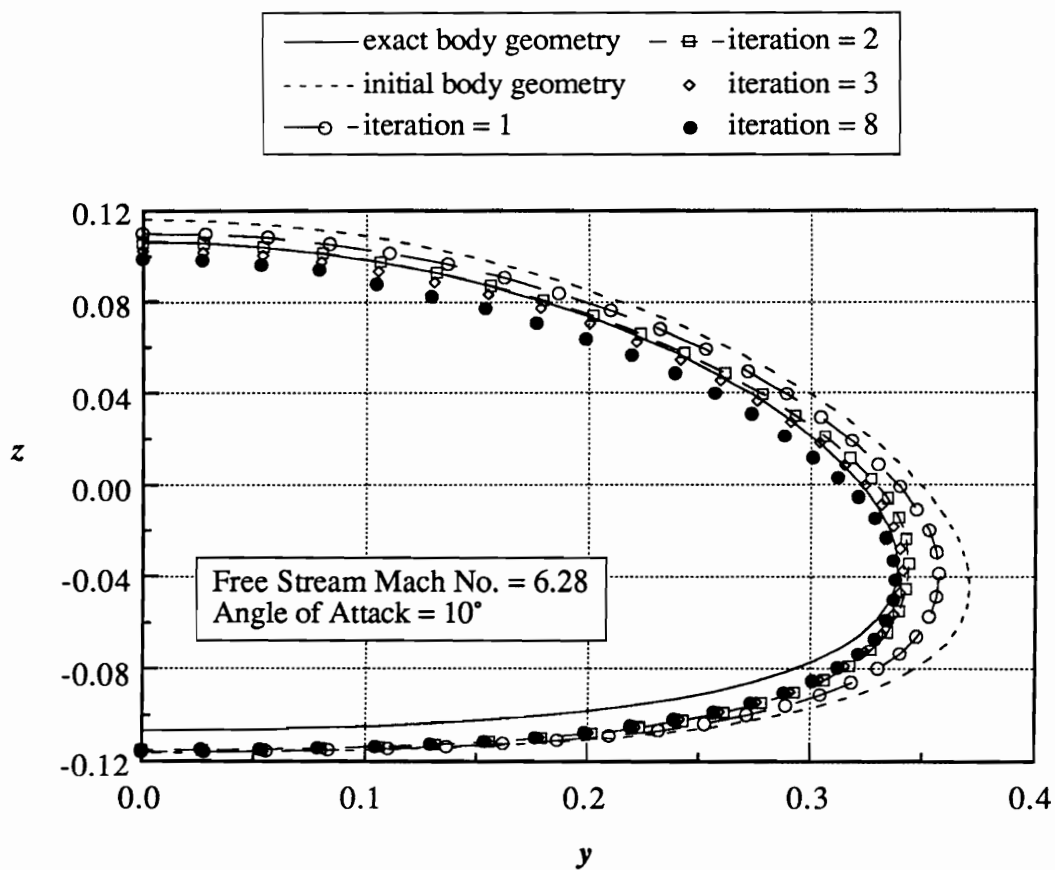


Figure 60. Inverse Solution Uniqueness: $M_\infty=6.28$, $\alpha = 10^\circ$
Resulting Body Geometries from Different Data Planes

TABLES

Table 1. Summary of the Various Minimum Drag Shapes

Name	Theory	Mach No. Range	Body Shape	$C_D(l/d)^2$
Karman Ogive	slender body	low - moderate supersonic	infinite slope and zero radius, at the nose zero slope at the base	1.0
Parker (1955)	linearized supersonic	low - moderate supersonic	pointed nose, nonzero body slope at the base	0.75 - 0.925
Eggers, et al. (1957)	Newtonian impact	∞	power law body $n = 0.75$	0.352
Hayes (1959)	Newtonian-Busemann	∞	power law body $n = 0.75$ initially and $1/3$ afterward	0.288
Cole (1957)	Newtonian-Busemann, slender body	∞	power law body $n = 0.66$	0.334
Fink (1966)	HSDT	hypersonic	depends on Mach no.	0.367 - 0.50

Table 2. Drag Coefficient of the Power Law Bodies with different Power n

		Inviscid (Euler) Calculation							PNS Calculation
		$l/d = 3.0$						5.0	3.0
Body Shape	M_∞ n	3.0	4.0	5.05	6.28	8.0	12.0	6.28	8.0
Power Law Shapes	0.650	0.067261	0.060736	0.056330	0.053017	0.049853		0.022727	$n = 0.69$ 0.059267 * 0.057683 **
	0.680	0.066999						0.022441	
	0.685		0.060266		0.052435				
	0.690	0.067025	0.060266	0.055830	0.052416	0.049267	0.045604	0.022404	
	0.695	0.067059	0.060275	0.055829		0.049260		0.022395	
	0.700	0.067064	0.060297	0.055840	0.052416	0.049266		0.022392	
	0.720				0.052555	0.049419			
	0.750	0.068193	0.061105	0.056539	0.053070	0.049987	0.046350	0.022617	
	1.000	0.082366	0.073550	0.068427	0.064906	0.061844		0.027168	
Hayes' Body	0.750				0.055011	0.052011	0.048431		

* Constant Wall Temperature Boundary Condition
** Adiabatic Wall Temperature Boundary Condition

Table 3. Comparison of Drag Coefficients between Perfect Gas and Equilibrium Air Model

M_∞	Inviscid Calculation (n = 0.75)		PNS Calculation (n = 0.69)	
	Perfect Gas	Equilibrium Air	Perfect Gas	Equilibrium Air
6.28	0.053070	0.053063 (0.013 %)		
8.00	0.049987	0.049981 (0.012 %)	0.057683*	0.057867* (0.32 %)
12.0	0.046350	0.046349 (0.0022 %)		

* Adiabatic Wall Temperature Boundary Condition
• Percent Differences are Given

Table 4. Convergence Behavior of Several Pressure - Shape Rules for Test Case I

Mach Number Pressure-Shape Rules	Mach number = 3.0		Mach number = 6.28	
	CPU(inverse)	No. of Iterations	CPU(inverse)	No. of Iterations
	CPU(direct)		CPU(direct)	
Linearized Supersonic	4.08	1 - 6	No Convergence	
Shock - Expansion	4.55	1 - 6	5.74	1 - 10
Tangent Cone	No Convergence		No Convergence	
Newtonian	No Convergence		No Convergence	
Linearized Supersonic with Regula-Falsi	3.49	1 - 3	3.71	2 - 4
Shock - Expansion with Regula-Falsi	3.54	1 - 4	3.91	1 - 4
Tangent Cone with Regula-Falsi	2.89	1 - 3	2.62	0 - 3

Table 5. Convergence Behavior of Several Pressure - Shape Rules for Test Case II

Mach Number Pressure-Shape Rules	Mach number = 3.0		Mach number = 6.28	
	CPU(inverse)	No. of Iterations	CPU(inverse)	No. of Iterations
	CPU(direct)		CPU(direct)	
Linearized Supersonic	4.36	1 - 6	No Convergence	
Shock - Expansion	4.36	2 - 5	4.45	0 - 5
Tangent Cone	No Convergence		No Convergence	
Newtonian	No Convergence		No Convergence	
Linearized Supersonic with Regula-Falsi	3.70	1 - 3	3.49	0 - 3
Shock - Expansion with Regula-Falsi	3.58	2 - 4	3.64	1 - 4
Tangent Cone with Regula-Falsi	2.28	0 - 3	2.09	0 - 2

Table 6. Convergence Behavior of Several Pressure - Shape Rules for Test Case III

Mach Number Pressure-Shape Rules	Mach number = 3.0		Mach number = 6.28	
	CPU(inverse)	No. of Iterations	CPU(inverse)	No. of Iterations
	CPU(direct)		CPU(direct)	
Linearized Supersonic	6.34	5 - 8	No Convergence	
Shock - Expansion	5.81	5 - 6	3.90	2 - 4
Tangent Cone	2.53	0 - 2	1.94	0 - 2
Newtonian	5.92	4 - 6	3.03	1 - 4
Linearized Supersonic with Regula-Falsi	3.81	3	4.80	3 - 8
Shock - Expansion with Regula-Falsi	4.28	3 - 4	3.86	2 - 5
Tangent Cone with Regula-Falsi	2.49	1 - 3	2.00	1 - 2

Table 7. Drag Coefficients of the Axisymmetric Inverse Test Cases

Free Stream Mach Number = 3.0

	Drag Coefficient	Max. Thickness	L / D
Testcase I	0.10625	0.09178	5.45
Testcase II	0.05845	0.09053	5.52
Testcase III	0.12440	0.19926	2.51
Testcase IV	0.04608	0.08638	5.79
Ogive Cylinder	0.14917	0.16666	3.0
3/4 Power body	0.06817	0.16666	3.0

Free Stream Mach Number = 6.28

	Drag Coefficient	Max. Thickness	L / D
Testcase I	0.20066	0.06413	7.79
Testcase II	0.07503	0.09254	5.40
Testcase III	0.12260	0.22127	2.26
Testcase IV	0.05083	0.08207	6.09
Ogive Cylinder	0.12500	0.16666	3.0
3/4 Power body	0.05305	0.16666	3.0

APPENDICES

Appendix A: Details of Analysis Code Application to the Minimum Drag Body Study

cfl3de was used for the aerodynamic design calculation performed for this study. With boundary conditions and computational grids which are suitable to the corresponding body shapes and the flow regimes, the code reliably provides converged solutions. The only difficult input selection is the value of the time step (or, CFL number). The value required to obtain convergence and rapid approach to the steady state solution depends on both the Mach number and the grid density. It is difficult to specify the best value for each calculation a priori. Basically this is a matter of experience. It is safe to start with smaller value of time step and then restart with larger value. Typical starting values of the time step range from 0.1 to 0.5 for the supersonic and hypersonic flow calculations.

cfl3de has a 'restart' option which can be used in both the space marching and the global iteration techniques. Near the nose where a small subsonic region exists, the global iteration technique was used. By restarting the global iteration solution near the nose, the remaining region (more than 99.9% of the entire body) was calculated using space marching technique (As mentioned in Chapter II, space marching technique was used after verifying that the solution neglecting the small subsonic region near the nose is nearly identical with the global iteration solution). The entire flow field is divided into several blocks. In each block, either space marching or global iteration technique can be selected to use and 1 ~ 3 restarts have been used to get a converged solution (usually the convergence criterion is 4 orders of magnitude decrease of the residual). Typically the entire body was divided into 2 ~ 3 blocks. In the block near the nose, 2 ~ 3 restarts (using the value from small time step near 0.1 to large time step near 1.0) were needed to get a converged solution. In the remaining parts of the entire body, only one restart with large time step was enough to get a converged solution.

Both the flux vector splitting scheme due to van Leer and flux difference splitting scheme due to Roe are available in the code. In most calculations, Roe's flux difference splitting scheme works very well, but for high speeds ($M_\infty > 10.0$) it is better to use van Leer's flux vector splitting scheme because a converged solution is not always obtained with Roe's scheme.

The **cfl3de** code has options to select first, second or third order spatial accuracy in i, j, k direction. It is recommended to start with first order spatial accuracy and use higher order accuracy after 1 ~ 2 order of magnitude decrease of the residual. This strategy does not work for the global iteration technique near the nose region which has very dense grids in a very small region ($x/l < 0.0004$). Thus first order accuracy was used until a converged solution was obtained.

Several options are available for the specification of boundary conditions. Boundary conditions in i, j, k direction are shown in figure A. Along the axial direction ($i0bc$) symmetry condition ($w = -w, v = -v$, where v and w are the velocity components in y, z direction) was used for the global iteration calculation near the nose. Either '*fixed to the reference conditions*' for nonconical bodies or '*use the solution at $i=1$ plane*' for conical shapes was used for the space marching calculations. Along the circumferential direction ($j0bc$ and $jdimbc$), either tangency condition (axisymmetric flow calculation) or symmetry condition ($v = -v$) for nonaxisymmetric bodies was used. In the body surface ($k0bc$), tangency condition was used for the inviscid calculation. No slip condition with either adiabatic wall temperature or fixed wall temperature was used for the viscous calculation (PNS). '*fixed to the reference conditions*' was used in the outer boundary ($kdimbc$). Either first order or second order extrapolation was used for $idimbc$.

Appendix B: Computational Grids Used in the Study

Several grid systems which are basically two dimensional grids have been generated and applied to the aerodynamic design calculations for this study. The implemented two dimensional grid generator¹ uses transfinite interpolation, elliptic grid generation and stretching. Transfinite interpolation is a multi-directional interpolation in which all the intersecting arbitrary curves within the boundary are considered to be interpolated⁹¹. This interpolation is a fast and efficient means of obtaining satisfactory grids for relatively simple geometries. The inputs for the grid generation using the transfinite interpolation are the grid point values at the boundaries (4 boundaries in two dimension). The specification of the boundary points is very important to obtain a good grid system which has maximum orthogonality between the neighboring cells. Because a grid system generated from this two dimensional grid generator is strongly dependent upon the specified boundary points, these points must be specified carefully. An elliptic grid generator solves a set of coupled elliptic partial differential equations to maximize the orthogonality of the grids for the relatively complex geometries. Because this routine requires an initial grid approximation, the transfinite grid generator runs first to provide initial grids. For the convergence criterion of 4 orders of magnitude decrease of the residual, converged grids are obtained within 300 ~ 400 iterations. *hyperbolic tangent* type stretching function was used to get a clustered grid near wall.

$$y_i = y_{k \dim} \cdot (1 - \eta_y) + y_1 \cdot \eta_y \quad (\text{B.1})$$

$$\text{where, } \eta_y = \beta \cdot \frac{e^{a_k} - 1}{e^{a_k} + 1} \quad (\text{B.2})$$

$$\text{and, } a_k = \left(\frac{k \dim - k}{k \dim - 1} \right) \cdot \log \left(\frac{\beta + 1}{\beta - 1} \right) \quad \text{for } k = 1, k \dim \quad (\text{B.3})$$

1. Program 'Grid2d' written by W.D. McGrory and R.W. Walters

Stretching factor β controls the amount of stretching. β of 1.02 ~ 1.03 are suitable for laminar calculation (β of 1.07 was used for the inviscid calculation of the study).

For the minimum drag body study, two different grid systems have been generated to calculate supersonic and hypersonic axisymmetric bodies. One is for the space marching portion of the computational domain and the other is for global iteration portion. Since power law bodies have an infinite slope at the nose, the shock is detached and hence there is a small subsonic region near the nose. This region depends on both free stream Mach number and power law index n . For $M_\infty = 6.28$, $n=0.75$, this region is less than 0.02% of the entire region. To capture this small region, very fine grid near the nose is required. In this region ($0 < x/l < 0.0004$), the global iteration technique is employed and the grid system of 52 points along the i direction, 25 points along the k direction, and 2 points along the j direction is used. For the remaining region which employs space marching ($0.0004 < x/l < 1.0$), the grid system of 41, 20, 2 points (typically) in i, k, j directions is used.

The boundaries of the axisymmetric grids used in the minimum drag body study are shown in figure B (the whole grids are shown in figure 4). The outer boundary of both the space marching portion and global iteration portion of the entire domain is specified by $Y_{outer} = c(x+a)^e$ where, c , a and e are constants and determined as follows: The outer boundary passes through specified points $(b, f\sqrt{r}b^n)$ and $(1.0, gr)$. r is the body radius at the base ($x/l = 1.0$). b is set to 0.0004 and a is set to $0.05 b$ for the power law body of $n=0.75$ at $M_\infty = 6.28$. Thus Y_{outer} is determined by specifying f and g values and can be adjusted with the corresponding free stream Mach number, by changing f and g values. Typical values for f and g are 2.0 and 3.0 for supersonic speeds ($M_\infty = 3.0$) and 1.5, 2.0 for hypersonic speeds ($M_\infty = 6.28$).

For three dimensional bodies which employ the space marching technique, the grid system which stacks two dimensional grids explained above, was used in each cross flow

plane. The outer boundary was specified by a circle, the radius of which was adjusted based on the free stream Mach number. Typical cross flow plane grids are shown in figure 47. For the elliptic cone (ratio between the major axis and minor axis, $a/b = 2.0$), $j_{max} \times k_{max}$ of 41×30 and 31×20 were used in each cross plane. The 'wing like' nonconical cambered body shape was generated from the elliptic cone of $a/b = 3.0$. First $3/4$ of the entire body shape is an elliptic cone and the planform shape of the remaining part of the body is specified as $3/4$ power law shape. In each cross plane, camber line is constructed by assuming a circular arc and specifying the camber slope angle at y_{max} point. This camber slope angle has cosine distribution from 0° at $x/l = 0.75$ to -12° at $x/l = 1.0$. New body geometry was obtained combining the camber line and the elliptic cone shape in each cross plane. 8 axial stations (6 stations for the nonconical parts of the body), which has $j_{max} \times k_{max}$ of 41×30 grids in each cross plane, were constructed. In order to have dense grids near the large gradient region (near y_{max} point), sine stretching ($y = a \sin(\pi/2 \times \eta)$, a is semispan length and $0 \leq \eta \leq 1$) was used both at the body surface and at the outer boundary.

Appendix C: About the Inverse Code

Minimum modifications have been made to the analysis code to develop the axisymmetric and three dimensional inverse methodology. In the analysis computer code, either a space marching problem or a global iteration problem is solved by sweeping the i station. The main subroutine for this sweep is subroutine '*goveq*' of the **cf13de**. Because the inverse methods developed step one cross plane at a time, using the same approach at each station, the entire inverse procedures are under the subroutine '*goveq*' and the code modifications are made inside this subroutine. For the axisymmetric inverse method, the entire procedure is included in the analysis code because the method is relatively simple and short. For the three dimensional inverse method, only the restart routine for inverse calculation, convergence evaluation, and regula-falsi root finding scheme are included in the analysis code. Target pressure distribution is also read in the subroutine '*goveq*'. The other modifications are added as new subroutines for the three dimensional inverse routine. In order to apply these inverse procedures to any existing computer code, common blocks and dimension statements are defined independently within the inverse routine.

The purpose of each subroutine is described as follows,

- ***initgr*** : By extrapolating the surface grid points in the previous cross planes this subroutine provides the initial grids at the body surface.
- ***crossgr*** : This subroutine generates the cross plane grids from the updated body geometry.
- ***invse3d*** : The inputs for this subroutine are the velocity components and the pressure distribution calculated from the analysis computer code using the updated body geometry. Surface streamlines and θ_0 are calculated from these

inputs. Subroutines '*norvec*', '*thetal*', '*y2z2*', and '*jspline*' are called to obtain a new body shape from the calculated surface pressure.

- ***norvec*** : This subroutine calculates a normal vector in each grid cell and then finds the surface inclination angle θ_a
- ***thetal*** : Using the given target pressure distribution, the calculated pressure distribution, local Mach number and θ_0 , new surface slope angle θ_1 in each grid point is calculated. Three surface pressure - body shape rules which were derived from supersonic/hypersonic shock-expansion theory, hypersonic tangent-cone theory and linearized supersonic theory are available in this subroutine.
- ***y2z2*** : Using the intersection points of the surface streamline to the current cross plane and θ_1 , a new body geometry is obtained in this subroutine. The subroutine follows the procedure described from equation (5.16) to (5.20).
- ***jspline*** : Using the updated body surface points, new grid points are obtained at the body surface by numerical interpolation and curve fitting. y_{max} and y_{min} points are obtained by 5 points least square parabolic curve fitting. For all the possible locations of y_{max} and y_{min} points, the method to index all the body geometry points has been incorporated in this subroutine. Using spline interpolation program the z components of the new grid points are obtained. Using this updated surface grid points, cross plane grids are generated at subroutine '*crossgr*'.

Three dimensional inverse subroutines described above are shown in the flow charts, figures 34 and 35.

Several control inputs for the inverse procedure as well as the target pressure distribution have been added to the **cfl3de** input file. Sample 'input' is included at the end

of this section. Even though the sample ‘input’ is shown for one cross plane calculation, inverse calculations of several cross planes can be performed at once. The control inputs for the inverse procedure are described as follows,

- *inverse* : specifies the starting plane of inverse calculation
- *iterinv* : specifies the maximum number of inverse iterations
- ΔC_{pmax} : convergence criterion (equation 5.1). 10^{-3} is used for most inverse calculations
- ΔC_p^2 : convergence criterion (equation 5.2). 5×10^{-4} is used for most inverse calculations
- *invmethd*: select surface pressure - body shape rule
 - = 1 supersonic and hypersonic shock expansion pressure - shape rule
 - = 2 hypersonic tangent cone pressure - shape rule
 - = 3 linearized supersonic pressure - shape rule
- *invrest* : restart the inverse solution
 - = 0 no restart
 - = 1 restart
- *ifalsi* : regula-falsi root finding scheme
 - = 0 no regula-falsi root finding scheme
 - = m use regula-falsi root finding scheme from m-th inverse iterations
where, m is any positive integer less than *iterinv*.

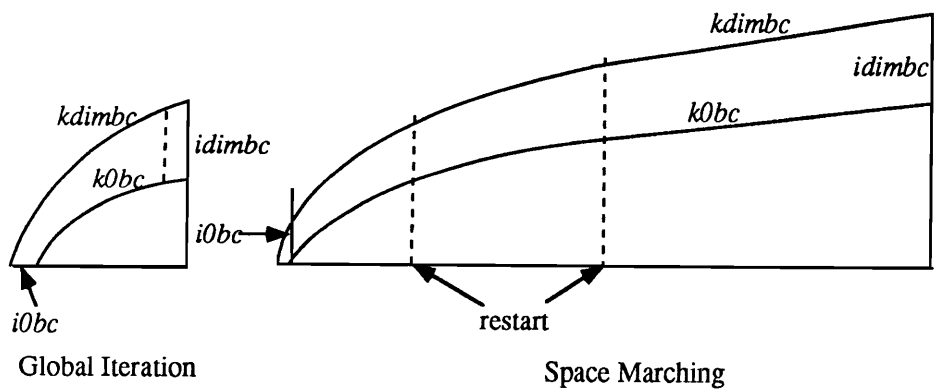


Figure A. Boundary Conditions in Power Law Body Study

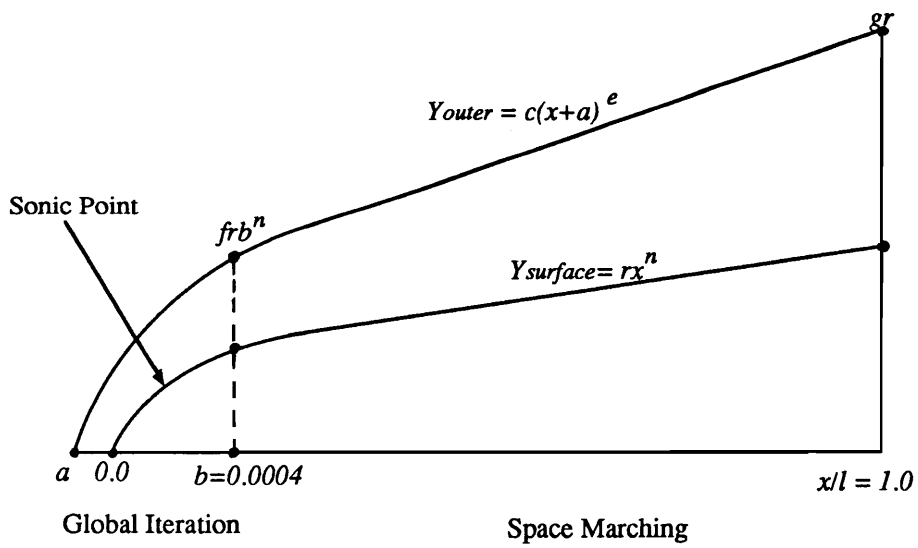


Figure B. Grid System for Power Law Body

```

NONCONICAL CAMBERED BODY ==== 3D TESTCASE III; ALPHA = 10 (DEG.)
ntzone
1
jmax(nz) kmax(nz) imax(nz) (have 1 to ntzone lines)
41      30      8
INVERSE ITER(MAX) DEL1INV DEL2INV INVMETHD INVREST IFALSI
7      10      1.E-3      5.E-4      1      0      0
xmach   re   alpha   pref   roref   tref   xlref   ivalue
6.28    0.45  10.0    1300.6  0.027313  100.0  1.0      1
igas    gamma  rgas    cpgas    prgas
1      1.4    286.9    1004.   .72
irest   dt     iflagts  fmax
1      -1.0    0      1.00
iroei   iroej   iroek    limi    limj    link    rkapi    rkapij    rkapi
1      1      1      2      2      2      0.3333    0.3333    0.3333
imarch  igrad   locmar   rtolp    initp    sigma
1      0      150    5.E-3    0      .950
irelu   nstlu   nrelu    iviscj   ivisk    iconsj   iconsk   ipturb
1      5      30      0      0      0      1      0
nit     locglo   rtolg     igrd     ioutpt   iplt3d   iform    ntzns
1      1      1.E-12    2      1      1      1      1
jte1    jte2     itel      ite2     istart   iend
1      41     1      8      7      7
i0bc    idimbc   j0bc      jdimbc   k0bc     kdimbc
2      1      2      2      2      1
TARGET PRESSURE INPUT;
-0.026267
-0.026067
-0.026112
-0.026747
-0.029909
-0.029956
-0.029403
-0.028674
-0.027629
-0.026291
-0.024740
-0.022923
-0.020987
-0.019154
-0.016711
-0.012522
-0.005058
0.006922
0.024722
0.052510
0.090311
0.136928
0.180399
0.213803
0.227146
0.223122
0.213269
0.200519
0.184662
0.166352
0.151780
0.139257
0.128082
0.118366
0.108231
0.100337
0.092713
0.089419
0.084467
0.083701

```

Sample 'Input' of the Inverse Calculation: 'Wing Like' Nonconical Cambered Body

$$M_{\infty} = 6.28, \alpha = 10^{\circ}$$

VITA

The author was born in Busan, Korea on January 1, 1962. He earned his B.S. degree in Aerospace Engineering at Seoul National University in February 1984, and earned his M.S. degree in Aerospace Engineering from Graduate School of Seoul National University in February 1986.

He fulfilled the military service as a second lieutenant. With two years experience in the industry (Hyundai Motor Company and Korea Advanced Institute of Science and Technology) as a research engineer, he started his Doctoral curriculum at the Aerospace and Ocean Engineering Department, Virginia Polytechnic Institute and State University in August, 1988.

His research interests are Aerodynamic Design, Applied Aerothermodynamics, Computational Fluid Dynamics and Numerical Methods, and Supersonic and Hypersonic Gas Dynamics.

He is currently a member of Korean Scientists and Engineers Association and a student member of AIAA.

A handwritten signature in black ink, appearing to be 'J. M. Lee', written in a cursive style.