DAHLGREN DIVISION
NAVAL SURFACE WARFARE CENTER
Dahlgren, Virginia 22448-5100

NSWCDD/TR-98/147

ZEUS^{++}—A GUI-BASED FLOWFIELD ANALYSIS TOOL, VERSION 1.0, USER'S MANUAL

BY DR. DAVID F. ROBINSON
WEAPONS SYSTEMS DEPARTMENT

FEBRUARY 1999

Approved for public release; distribution is unlimited.

DESTRUCTION NOTICE—For unclassified, limited distribution documents, destroy by any method that will prevent disclosure of contents or reconstruction of the document.

REPORT DOCUMENTATION PAGE

Form Approved
OBM No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)

2. REPORT DATE
February 1999

3. REPORT TYPE AND DATES COVERED

4. TITLE AND SUBTITLE
Zeus^{++}—A GUI-based Flowfield Analysis Tool, Version 1.0, User's Manual

5. FUNDING NUMBERS

6. AUTHOR(s)
Dr. David F. Robinson

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Naval Surface Warfare Center
Dahlgren Division (Code G72)
17320 Dahlgren Rd
Dahlgren, VA 22448-5100

8. PERFORMING ORGANIZATION REPORT NUMBER
NSWCDD/TR-98/147

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

A User's Manual for the Zeus^{++} Flowfield Analysis tool is provided along with a tutorial for setting up and executing a missile geometry with control surfaces. The tool solves the three-dimensional Euler equations to calculate the inviscid flowfield about the geometry. This inviscid solution is then supplied as input to an integral boundary layer solver which calculates the effects of viscosity on the body. Finally, if a more in-depth solution is desired, Zeus^{++} can be used to generate a computational domain for use in a full Navier-Stokes solver.

By applying the Zeus^{++} tool, computational analyses of complex three-dimensional missile geometries can be performed on a personal computer in a matter of minutes. In addition, a computational grid suitable for a full Navier-Stokes solver can be generated in approximately 10 min. This represents significant savings in both manpower and computational costs typically associated with computational fluid dynamics.

14. SUBJECT TERMS
Zeus^{++} computational fluid dynamics (CFD) GUI-based Flowfield

15. NUMBER OF PAGES
96

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	**18. SECURITY CLASSIFICATION OF THIS PAGE**	**19. SECURITY CLASSIFICATION OF ABSTRACT**	**20. LIMITATION OF ABSTRACT**
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	SAR

FOREWORD

The current work is in response to the demand for a computational tool that can be used for rapid and inexpensive aerodynamic analysis of missile-type geometries. Although it is possible to evaluate missile geometries using a full Navier-Stokes computational fluid dynamics procedure (CFD), both budgetary and time constraints often preclude this type of in-depth analysis. The Zeus^{++} Flowfield Analysis Took was developed in order to minimize the cost and turnaround time associated with aerodynamic analysis of missile geometries. The tool can be used to either perform a flowfield analysis, or alternatively, to generate computational domains for a full Navier-Stokes solver.

Initial funding for the work was provided by the Modeling and Simulation Task of the Air and Surface Weaponry Technology Program funded and managed by the Office of Naval Research, Code 351. Additional funding was provided by George Long through the Standard Missile Program, PMS422. The author would like to express his appreciation to both Gil Graff and George Long for their support during this project.

This report has been reviewed by Mr. Thomas A. DelGuidice, Head, Weapons Integration and Technology Branch; and Mr. Bob Steigler, Head, Combat Systems Safety and Engineering Division.

Approved by:

JOHNNY WALTERS, Deputy Head
Weapons Systems Department

CONTENTS

Section Page

ILLUSTRATIONS

CHAPTER 1: INTRODUCTION

1.1 OVERVIEW OF ZEUS^{++}

In the past decade, significant advances have been made in the field of computational fluid dynamics (CFD). These include improvements in computational speed, numerical algorithm efficiency, and modeling (turbulence and combustion). Because of this progress, it is now commonplace to see accurate CFD solutions to very complex, large scale, three-dimensional flowfields. It should be noted, however, that a number of limitations remain. First, existing CFD solvers are quite complex and extensive training is required prior to generating a solution. The user must be trained not only in basic aerodynamics, but also in computational domain development and numerical methods. Second, the computational costs are still extensive. Complex cases can still take days, weeks, or even months of expensive workstation or supercomputer time. Because of these limitations, parametric type design studies are often infeasible, and therefore, the use of CFD as an engineering design/development tool has been limited.

The goal of the current work is to significantly reduce both the costs and turnaround time associated with CFD solutions of missile type geometries with arbitrary control surfaces. The current work began with the ZEUS flowfield solver[2,4,7,8]. The ZEUS code is a supersonic space marching Euler solver capable of computing both internal and external flowfields. The current work involved modifying the ZEUS code to make it more amenable to solving for the forces/moments on missile type geometries with arbitrary control surfaces. A number of limitations in the original code motivated the current work:

1) For each geometry under consideration, the user must develop a Fortran source code that provides a geometric description of the missile surface. In addition, code that specifies the control surfaces and switching logic for the boundary conditions must be generated.

2) The user must be familiar with a number of separate codes in order to implement a ZEUS run.

These include a separate code for each of the following tasks:

a) Cone/Blunt - Generate an initial profile by running one of the two codes.
b) Convert - Convert the initial profile to a form readable by ZEUS
c) ZEUS - Solves the Euler form of the equations of motion.
d) BL - Performs a boundary layer analysis of the inviscid flowfield.

Since the user must understand all of the inputs/outputs from each of the above codes, this significantly increases the amount of training required.

3) The setup time required before executing the code can be significant (typically on the order of hours/days). In addition, if parametric studies are desired, a separate set of input files must be generated for each case under consideration. This is both a time consuming and error prone process when a large number of cases are desired.

4) Inherent limitations (space marching) in the ZEUS code prevent the solution of any flowfield with subsonic regions.

The modifications made to the ZEUS code focused on the following four primary areas of improvement:

1) Minimize the required training - This was done by developing a graphical user interface, modifying the grid generation capabilities, and developing simple to use pre- and post-processing features. The GUI allows for most of the complexity of the code to be hidden from the user. Instead of developing Fortran modules to specify the missile and control surface geometries, the current tool (Zeus^{++}) uses a series of point-and-click GUI menus to setup the required geometry files. All of the complexities of generating a three-dimensional grid and blending control surfaces to the missile have been hidden from the user. The GUI interface also significantly simplifies the post-processing of the results.

2) Reduce the setup/execution time - The interface reduces the setup time required to a few minutes vs. many hours/days for the original ZEUS code. In addition, the user can setup the code to sweep through a variety of parameters (Mach number, angle of attack, angle of yaw, and roll angle) by simply specifying an initial, final, and incremental value for each of the parameters. All of the complexity involved with generating numerous input files has been hidden from the user. This feature allows for parametric type studies with no additional setup time required.

3) Minimize the costs - The costs associated with computational fluid dynamic solutions are often prohibitively high, and therefore, prevent the application of CFD during the design/analysis phase of missile development. The two primary influences on these costs are personal and computational requirements. The personal costs are inflated because you must have someone with extensive training performing a very time consuming task. Zeus^{++} reduces the personnel costs by minimizing the required training and reducing the time required to setup a run. The second factor, computational cost, is high because the ZEUS code was developed for use on expensive Unix workstations. These costs were reduced by porting the ZEUS code to less expensive personal computers and by developing a GUI that runs under a widely available operating system (Windows NT and Windows 95).

4) Extend the range of applications for the code - There are a number of cases where either the ZEUS code cannot be applied, or where the results may be suspect due to the approximate nature of the solution procedure (space marching Euler). For these cases, the Zeus^{++} tool includes a grid generator which will

output a three-dimensional grid and the appropriate boundary conditions which can then be applied as input for the GASP[1] full Navier-Stokes flowfield solver. It should be noted that generation of complex three-dimensional missile grids with control surfaces often took days to develop. The Zeus^{++} tool reduces this to a matter of minutes on a personal computer.

1.2 DOCUMENTATION

The documentation for the Zeus^{++} tool is presented in one of two forms: online help or a printed user's manual. It should be noted that the data contained within the two forms are identical.

To access the online help press the 'F1' key during execution of the Zeus^{++} code. If the user does not understand one of the requested inputs, pressing the 'F1' key will display the help file for that specific topic. Alternatively, the user can search through the help documentation online by clicking on 'Help' from the main view screen.

A hard copy of the Zeus^{++} user's manual can be obtained by executing the code (double clicking on ZeusPP.exe) and selecting "Print User's Manual". Alternatively, the manual may be printed by selecting "Help" and then "Print User's Manual" from the main window.

> Note: The manual is provided in Adobe pdf format. The Adobe Acrobat reader may be obtained from Adobe's home page at "http://www.adobe.com/acrobat". Alternative forms of the User's manual can be supplied upon request if the pdf format cannot be viewed/printed.

1.3 SYSTEM REQUIREMENTS

Processor:	Intel 486 compatible (233 MHz Pentium II or higher recommended)
Operating System:	Microsoft Windows NT 4.0 (recommended), Windows 95, or Windows 98
Plotting Package:	Tecplot v7.0 (Optional)
Hard Disk Space:	Approximately 3 Megabytes for the Zeus^{++} code and associated files. The size of the output data generated by Zeus^{++} is highly case dependent (grid resolution, # of sweeps, etc…) and can range from a few megabytes to a few hundred megabytes.

1.4 CONTACT THE AUTHOR

E-mail requests for a copy of the code to the author at the following:

Dr. David F. Robinson (Code G72)
Naval Surface Warfare Center
17320 Dahlgren Rd.
Dahlgren, VA 22448-5100
robinsondf@nswc.navy.mil

Voice: (540) 653-3954
Fax: (540) 653-8268

1.5 EXECUTE THE CODE

Execution of the code begins by double clicking on the program 'ZeusPP.exe', entering a name for the current case, and then clicking on 'start New Case'.

Note: 1) The name entered cannot contain blank spaces.

2) A tutorial for setting up and executing a new case is provided in Appendix A.

3) The code must be executed from the main ZeusPP directory. To execute one of the sample cases provided, copy the file from the 'samples' directory to the ZeusPP main directory.

To open an existing case (*.zpp) click on 'ZeusPP.exe' and select 'Open Existing Case'. From the main menu click on 'File/Open' and select the desired case. Alternatively, simply double click on the file ('filename.zpp').

CHAPTER 2: MAIN WINDOW CONTROLS

2.1 GEOMETRY SKETCH

As the geometry of the missile and its control surfaces are specified, two-dimensional sketches of the side and frontal views are displayed in the main view screen (see example below). There are a variety of methods for manipulating the sketch, each of which is listed in the upper left corner of the main view screen and discussed in detail below.

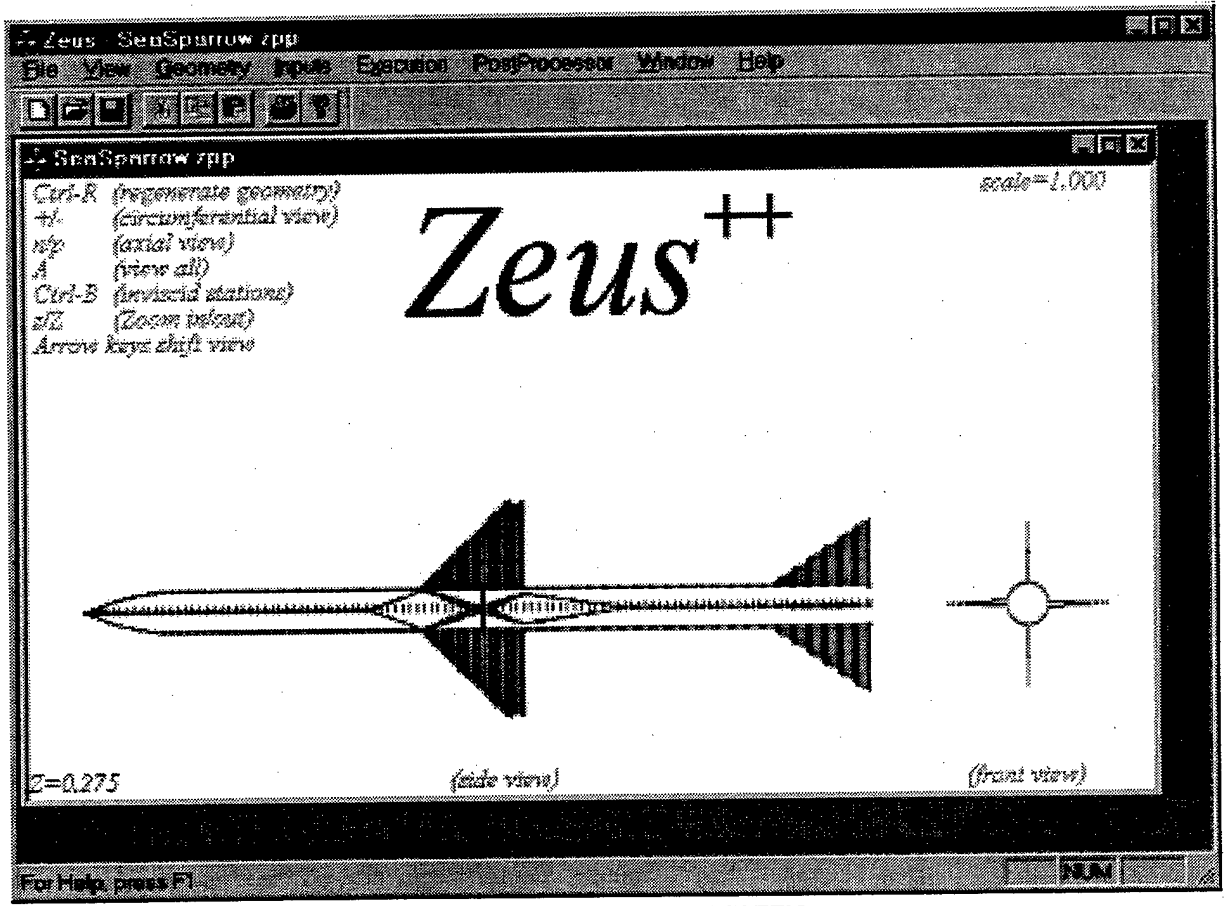


FIGURE 1. MAIN VIEW SCREEN

- *Ctrl-R* - Pressing the 'Control' and 'R' keys simultaneously will regenerate the geometry display shown in the main window. This should be done any time the missile geometry is modified and an updated view is desired. Note that this view is for display purposes only, and therefore, to limit computational effort, a low-resolution representation of the surface geometry is provided. This is not representative of the computational domain employed during the solution procedure. If desired, the resolution of the display can be increased/decreased by modifying the number of points in the View/Options section.

- *+/-* - Pressing the '+' and '-' keys steps the views circumferentially through each of the zone edges. '+' shows the next and '-' shows the previous circumferential view.

- *n/p* - Pressing the 'n' and 'p' keys steps the front view axially down the missile. 'n' and 'p' go to the next and previous axial location, respectively. Pressing 'N' or 'P' has the same effect, however, larger axial steps are taken. The current axial location is shown in the lower left corner of the main view screen (z=0.275).

- *A/a* - Pressing the 'A' or 'a' key redraws all of the axial stations and circumferential planes.

- *Ctrl-B* - Pressing the 'Control' and 'B' keys simultaneously will display the axial stations where the inviscid grid was output. These stations are shown as blue tic marks along the centerline of the missile geometry. *This feature cannot be employed until after an inviscid run has been completed.*

Note: The viscous boundary layer solver requires the inviscid Euler solution as a boundary condition. Therefore, the inviscid solution must be performed before running the viscous code. To obtain an accurate viscous approximation an adequate number of inviscid profiles must be provided such that all relevant geometric features are spatially resolved. The inviscid profiles are written out at the user-specified intervals provided in the boundary layer inputs section.

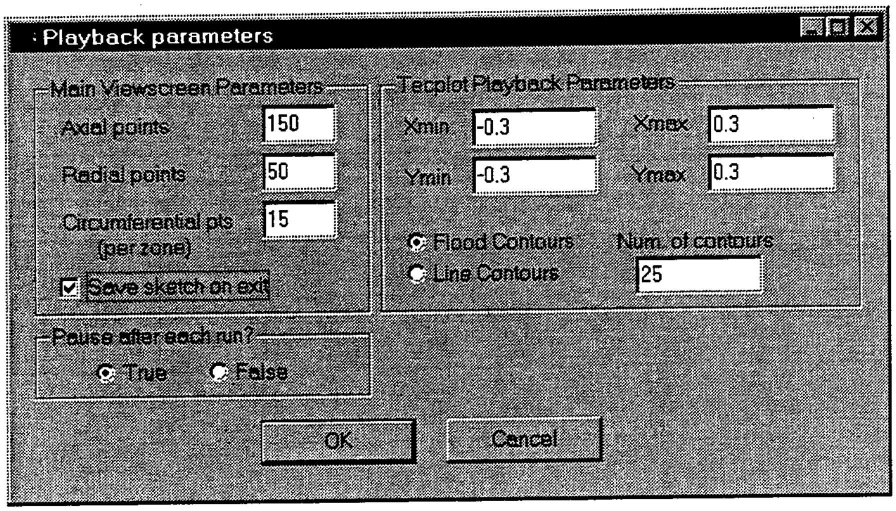
- *'z'/'Z'* - Pressing the 'z' or 'Z' keys zooms in/out on the main view screen. The amount of scaling applied to the original sketch is shown in the upper right hand corner. To return to the original settings, press Ctrl-R to regenerate the sketch.

- *Arrow keys* - The arrow keys are used to translate the sketches in the main window.

2.2 MENU OPTIONS

File	Controls file I/O and printing.
View	Toolbar views and runtime options.
Geometry Setup	Define the missile and control surface geometries as well as the computational domain.
Inputs	Input free-stream conditions, runtime parameters, reference conditions, ...etc.
Execution	Executing the code
Post-Processor	Post-Processing
Window	Layout and orientation of windows
Help	Help controls and 'About' information (author, version #, and contact information)

2.3 MISCELLANEOUS OPTIONS



Axial points 150
Radial points 50
Circumferential pts (per zone) 15
☑ Save sketch on exit
Xmin -0.3
Xmax 0.3
Ymin -0.3
Ymax 0.3
● Flood Contours
○ Line Contours
Num. of contours 25
● True ○ False

FIGURE 2. PROGRAM CONTROL PARAMETERS

- *Number of Points* - The sketch shown in the main window is a low-resolution representation of the current missile geometry. To increase/decrease the resolution of the geometric display simply increase/decrease the appropriate points parameter (axial, radial, or circumferential). Increasing the resolution will increase both the time to display the sketch as well as the size of the output data file (*.zpp).

- *Save sketch* - The output data file for the Zeus^{++} code (*.zpp) contains all of the input parameters as well as the coordinates for the geometry sketch shown in the main window. To minimize the size of the output files (*.zpp) de-select the 'save sketch on exit' dialog box shown above. This will significantly decrease the size of the output file (approximately 3k vs. 1-2 meg.); however, when the case is restarted, the geometry sketch must be regenerated. If the size of the output file is not important then it is recommend that this option be selected to minimize the computational effort required at startup.

- *Tecplot Parameters* - A number of Tecplot macro files (contour plots, movie files, ...etc.) are generated by the Zeus^{++} code in order to simplify post-processing. The following parameters, used in the macro files, are specified in the View/Options section.

  X_{min}, X_{max} - The minimum/maximum x-axis range for the movie files.
  Y_{min}, Y_{max} - The minimum/maximum y-axis range for the movie files.
  Flood/Line - Specify either flood or line drawing for the contour plots.
  # of contours - Specify the number of contour lines to be drawn.

- *Pause* - Zeus^{++} runs a number of external programs from the command prompt. If the 'Pause after each run' feature is selected then the command prompt will remain active (i.e., paused) and prompt the user to 'press any key' before exiting. If the pause feature is false, the Zeus^{++} code will automatically close the command prompt upon termination of the code.

CHAPTER 3: GEOMETRY SETUP/EVALUATION

MENU OPTIONS

Grid Parameters	Specify the grid parameters for both the Euler and Navier-Stokes grids. (# of grid points, clustering, # of zones, and location of outer boundary).
Missile Sections	Define the missile geometry
Control Surfaces	Define the control surface geometry
Grid Generation	Generate the Euler and Navier-Stokes computational domains.

3.1 COMPUTATIONAL DOMAIN

3.1.1 Grid Parameters

Recall the Zeus^{++} tool has two primary functions. The code serves not only as an approximate flow solver, but also as a Navier-Stokes grid generator. Therefore, two separate sets of grid parameters are specified when setting up the computational domain. In the first, the user inputs the parameters for the computational domain that will be used during the Euler and boundary layer solutions of the flowfield. In the second, the input parameters are used to generate the Navier-Stokes grid and appropriate boundary conditions.

Mesh Dimensions

- *Number of zones* - The number of zones is equal to the maximum number of control surfaces at a given axial location (i.e., if you have 4 canards, 3 dorsal fins, and 5 tails fins then you must enter '5' as the number of zones).

- *No control surfaces* - Select this box if you are considering a case with no control surfaces.

- *Number of radial cells* - The number of cells located radially between the body surface and the outer boundary of the computational domain.

- *Default number of circumferential cells/zone* - Changing this parameter will change the number of circumferential cells/zone for all zones.

- *Number of cells/zone* - The number of cells/zone refers to the circumferential direction (i.e., between zone edges) and does not have to be the same for all zones. Change an individual zone by clicking on the desired zone and entering a new value.

Mesh Clustering

The flowfield solver typically does not require mesh clustering because there is no boundary layer to resolve. However, in certain cases, mesh clustering (radial and circumferential) can be used to more accurately capture specific flowfield features (vortices, shocks, etc). Note that mesh clustering is required for the Navier-Stokes grid in order to resolve the boundary layer.

- *Radial clustering* - Specify the grid clustering between the body surface and the outer boundary. A value of 1.0 provides equal spacing and a value of 0.0 provides maximum clustering at the surface.

  Note: 1) Two sets of radial clustering are required. These correspond to the radial clustering at the initial and final (nose and tail) axial stations. This provides the ability to have more radial clustering at the nose than at the tail of the missile, or vice-versa. The radial clustering parameter is interpolated linearly between the two values as you progress axially down the missile.

  2) Clicking the 'Boundary Layer Thickness' button provides an estimate of the boundary thickness (laminar and turbulent) at a given axial station. The values are calculated using flat plate boundary layer assumptions and can be used to estimate the amount of radial clustering required for a viscous grid. This calculation requires a unit Reynolds number and an axial location. Initial spacing for the viscous grid is typically taken as 1/30th of the estimated boundary layer thickness.

- *Circumferential clustering* - Specify the clustering at the zone edges. A value of 1.0 provides equal spacing and a value of 0.0 provides maximum clustering at the zone edges (i.e., circumferentially between control surfaces).

Outer Boundary

- *Distance to outer boundary* - If the outer boundary of the computational domain is not tracking a shock (i.e., Free-Stream starting condition) then the radial distance from the body surface to the outer boundary is specified in terms of body radii (typically 10-20 body radii for an inviscid solution).

- *Growth of outer boundary* - In certain cases it is desirable for the radial distance between the body surface and the outer boundary to increase as you march axially down the missile. This parameter specifies the angle (in degrees) which the outer boundary grows relative to the body surface.

  Note: The above parameters are used during the Navier-Stokes grid generation. They are also used during the Euler solution if the outer boundary does not track the conical/bow shock. The type of outer boundary applied by the flowfield solver is specified in the Integration Parameters section.

Copy Parameters

- *Select 'Copy Parameters' button to copy the grid parameters from the alternate grid (i.e., from the Navier-Stokes grid if you are in the Euler grid setup, or vice-versa)*

3.2 MISSILE GEOMETRY

3.2.1 Select Geometry Type

The first step in setting up the geometry is to define all of the sections that comprise the missile surface (i.e., not control surfaces). First, select the desired type of nose and then click on 'Add New Section'. You will be prompted for the geometric parameters used to generate the desired type of nose section. After the nose section has been specified, repeat the above process for any remaining sections (i.e., afterbody, boat-tail, etc…). Note that the geometry sections must be entered in the order in which they occur on the geometry.

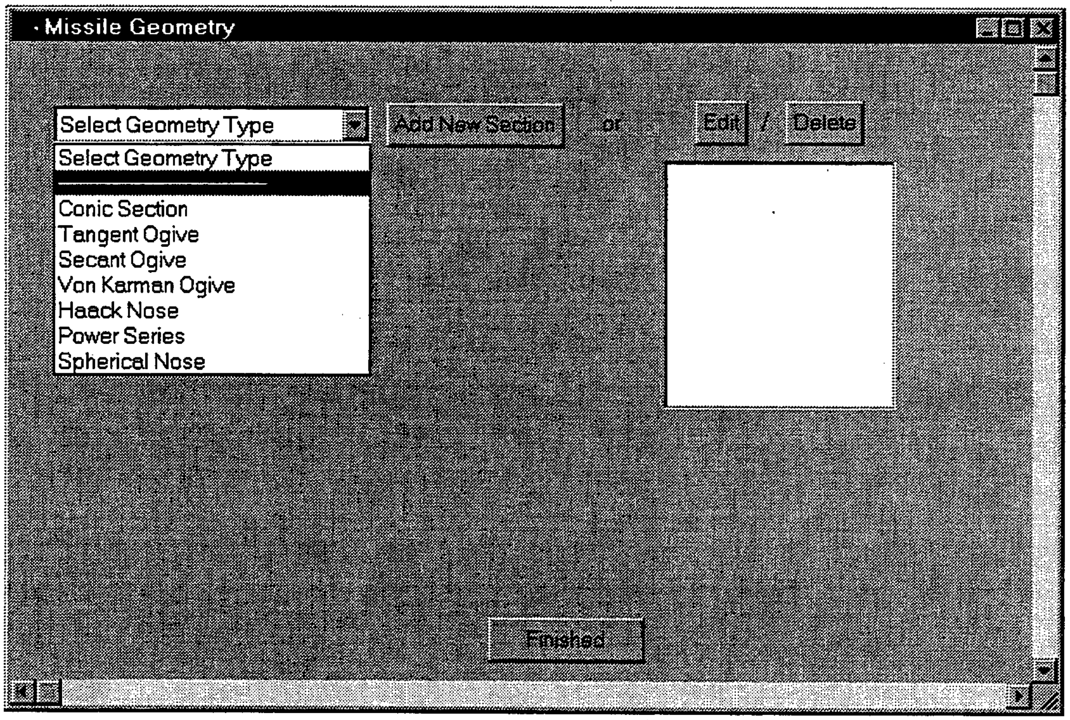
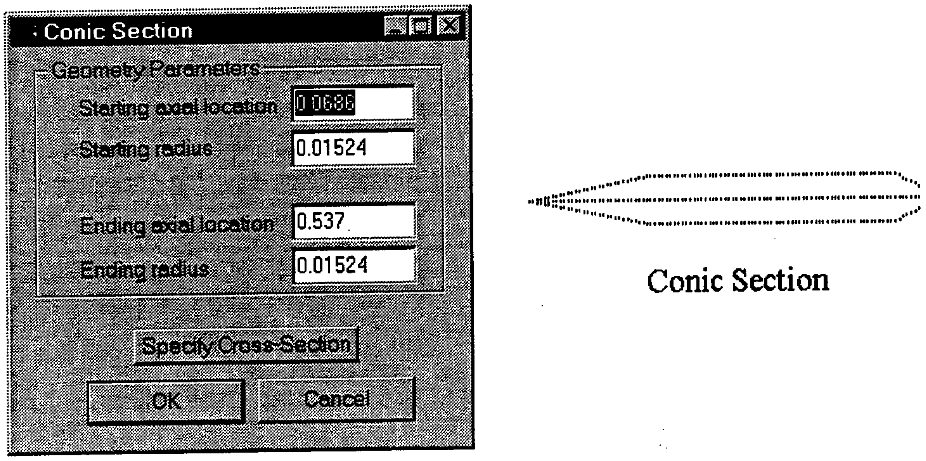


FIGURE 3. MISSILE GEOMETRY SELECTION

It should be noted from the above figure, that there are a variety of possible missile sections. A sample sketch, a description of the required parameters, and the generating equations are shown below for each of the available geometry sections. In all of the geometric formulas discussed below, 'z' is the axial distance from the start of the given section.

3.2.2 Conic



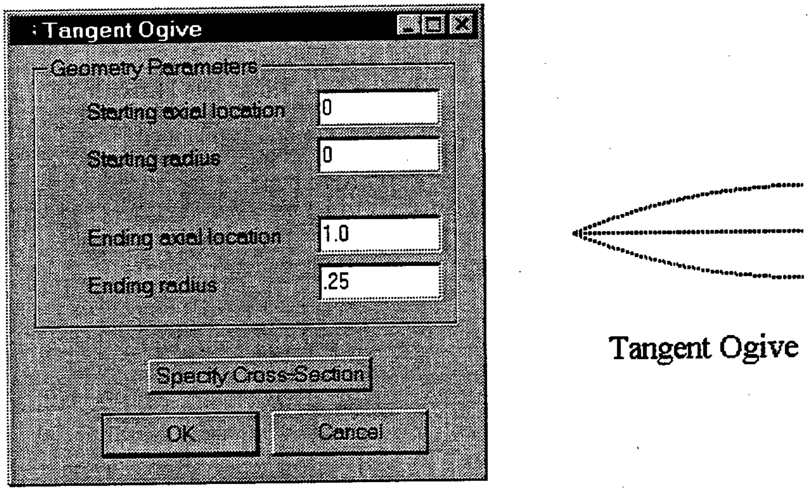
Geometry Parameters
Starting axial location 0.0686
Starting radius 0.01524
Ending axial location 0.537
Ending radius 0.01524

Conic Section

FIGURE 4. CONIC SECTION PARAMETERS

A variety of geometries can be described using the conic input section. These include those shown in the sketch above, namely, a nose cone, a cylindrical section, and a boat-tail flare. The parameters required to generate a conic section are shown in the above figure.

3.2.3 Tangent Ogive



Geometry Parameters
Starting axial location 0
Starting radius 0
Ending axial location 1.0
Ending radius .25

Tangent Ogive

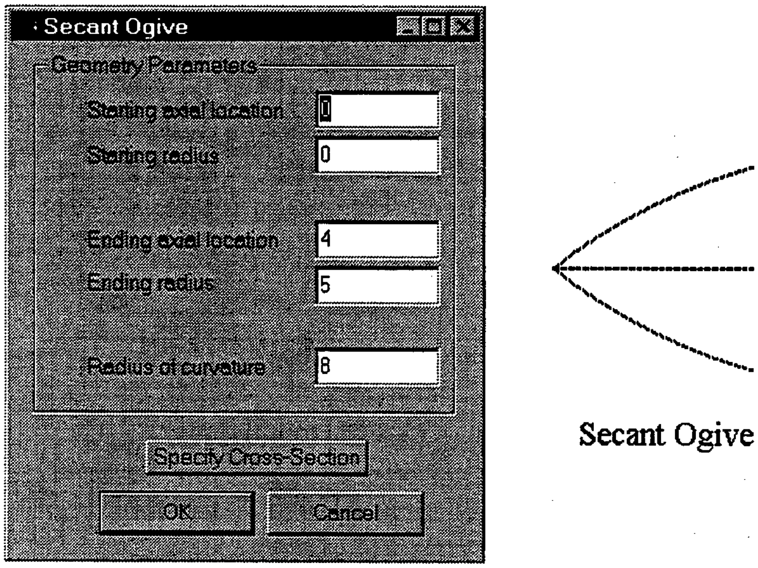
FIGURE 5. TANGENT OGIVE PARAMETERS

A sample sketch of a tangent ogive is shown above along with the parameters required to generate the section. Note from the sketch that the missile section is tangent to the horizontal at the aft axial station, hence the name, tangent ogive. The geometry is generated using the following formula[10] along with the user specified parameters R and L, where R is the base radius and L is the length of the section.

$$r = \sqrt{\rho^2 + (z - L)^2} + (R - \rho)$$

$$\rho = \frac{R^2 + L^2}{2R}.$$

3.2.4 Secant Ogive



Secant Ogive

Geometry Parameters

Starting axial location: 0

Starting radius: 0

Ending axial location: 4

Ending radius: 5

Radius of curvature: 8

Secant Ogive

FIGURE 6. SECANT OGIVE PARAMETERS

A sample sketch of a secant ogive is shown above along with the parameters required to generate the section. The geometry is generated using the following formula[10] along with the user specified parameters L, R, and ρ, where L is the length of the section, R is the base radius, and ρ is the radius of curvature.

$$r = \sqrt{\rho^2 + (z - h)^2} + k$$

$$h = \rho \cos\alpha$$

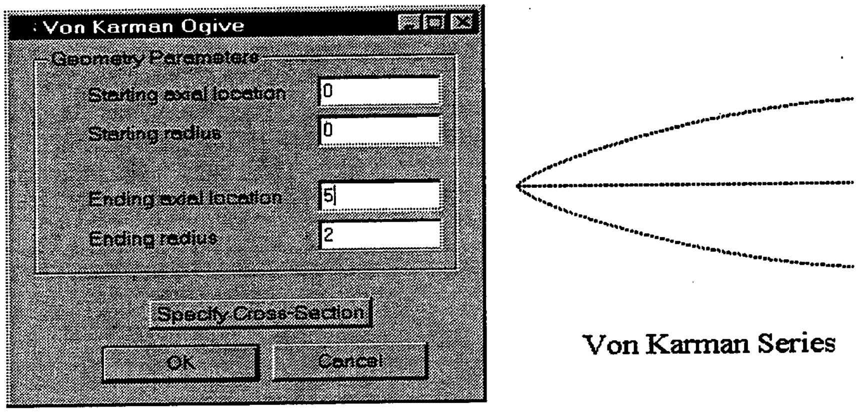
$$k = \rho \sin\alpha$$

$$\alpha = \gamma - \theta$$

$$\theta = \tan^{-1}\left(\frac{R}{L}\right)$$

$$\gamma = \cos^{-1}\left(\frac{\sqrt{L^2 + R^2}}{2\rho}\right)$$

3.2.5 Von Karman Ogive



Starting axial location 0
Starting radius 0
Ending axial location 5
Ending radius 2

FIGURE 7. VON KARMAN OGIVE PARAMETERS

A sample sketch of a Von Karman ogive is shown above along with the parameters required to generate the section. The geometry is generated using the following formula[10] along with the user specified parameters R and L, where R is the base radius and L is the length of the section.

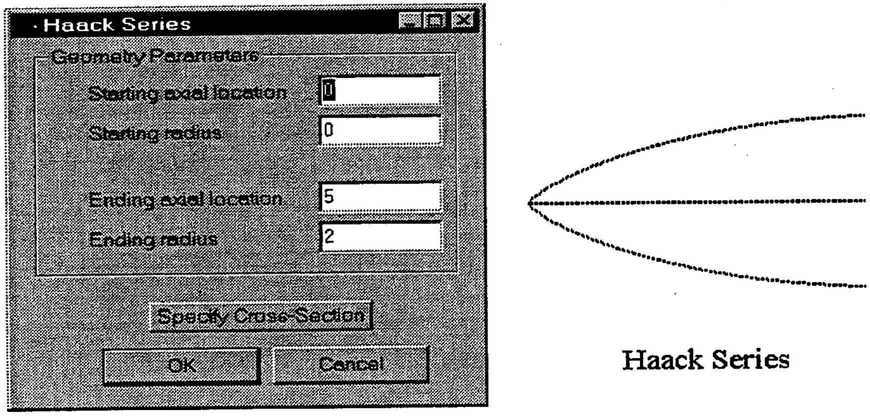
$$\bar{r} = \sqrt{\frac{\left(\phi - \frac{\sin(2\phi)}{2} + C\sin^3\phi\right)}{\pi}}$$

$$\phi = \cos^{-1}(1 - 2\bar{z})$$

$$C = 0$$

$$\bar{r} = \frac{r}{R}, \quad \bar{z} = \frac{z}{L}$$

3.2.6 Haack Series



Starting axial location 0
Starting radius 0
Ending axial location 5
Ending radius 2

FIGURE 8. HAACK SERIES NOSE PARAMETERS

A sample sketch of a Haack series is shown above along with the required input parameters. The geometry is generated using the following formula[10] along with the user specified parameters R and L, where R is the base radius and L is the length of the section.

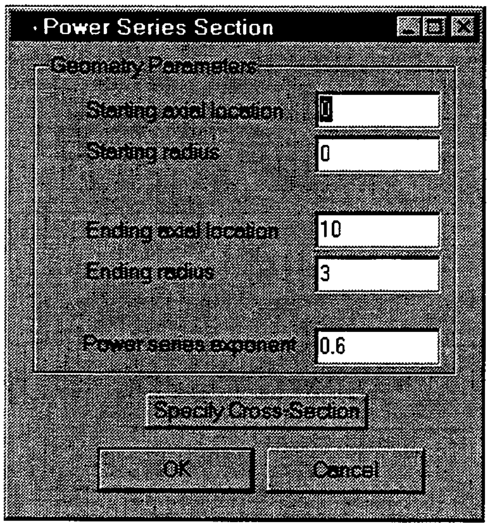
$$\bar{r} = \sqrt{\frac{\left(\phi - \frac{\sin(2\phi)}{2} + C\sin^3\phi\right)}{\pi}}$$

$$\phi = \cos^{-1}(1 - 2\bar{z})$$

$$C = \frac{1}{3}$$

$$\bar{r} = \frac{r}{R}, \quad \bar{z} = \frac{z}{L}$$

3.2.7 Power Series



Power Series Section

Geometry Parameters

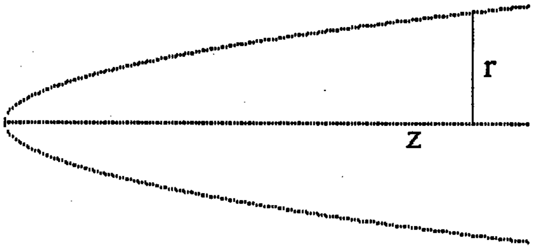
Starting axial location: 0

Starting radius: 0

Ending axial location: 10

Ending radius: 3

Power series exponent: 0.6



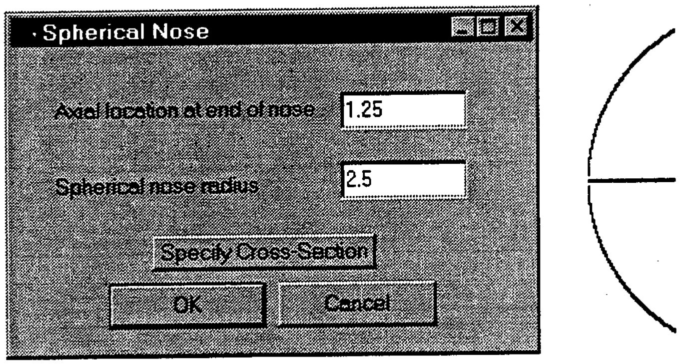
Power Series

FIGURE 9. POWER SERIES NOSE PARAMETERS

A sample sketch of a Power series section is shown above along with the required input parameters. The geometry is generated using the following formula[10] along with the user-specified parameters R, L, and n, where R is the base radius, L is the length of the section, and n is the power series exponent.

$$r = z^n$$

3.2.8 Spherical Nose



Axial location at end of nose 1.25

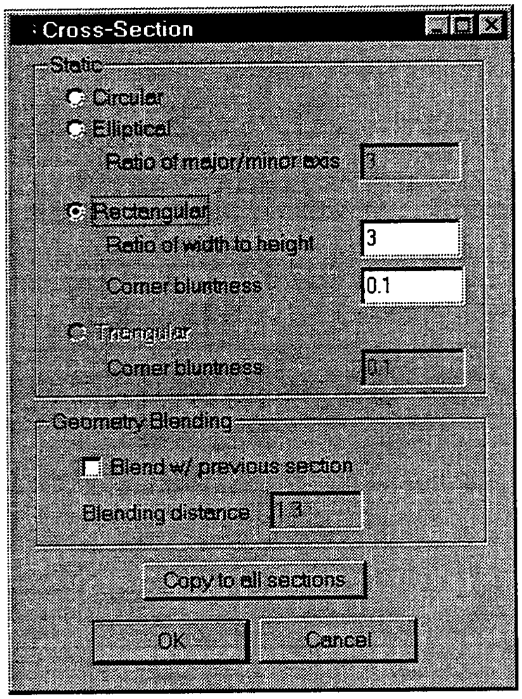
Spherical nose radius 2.5

FIGURE 10. SPHERICAL NOSE PARAMETERS

A sample sketch of a spherical nose section is shown above along with the parameters required to generate the section. For the spherical nose, there will be a small region where the flow is subsonic. Because the Euler solver employs a marching procedure, the Zeus code cannot be applied in this subsonic region. Therefore, the blunt body solver must be applied over the entire subsonic region. The outflow plane from the blunt body solver can then be taken as the inflow plane for the Euler solver. The input parameters for the blunt body solver are specified in the Integration Parameters section.

3.2.9 Cross-Sectional Geometry

For each missile section defined, a cross-sectional geometry must be specified. If a cross-section is not designated the default value of 'circular' will be used.



○ Circular

○ Elliptical

Ratio of major/minor axis 3

● Rectangular

Ratio of width to height 3

Corner bluntness 0.1

○ Triangular

Corner bluntness 0.1

☐ Blend w/ previous section

Blending distance 1.3

FIGURE 11. MISSILE CROSS-SECTION PARAMETERS

- *Cross-Section* - Select the desired cross-sectional geometry.
- *Corner Bluntness* - Specify a value between zero (no blunting) and 0.999 (completely blunted corners) for the Rectangular cross-sectional geometry.
- *Blending* - It is often desirable to have varying cross-sectional geometries for different sections of the missile (i.e., circular nose cone and elliptical afterbody). For these cases, the cross-sectional geometry cannot be abruptly changed, but rather, must be blended axially between the two sections. If blending is desired, then select the 'Blend w/ previous section' box and specify a blending distance in ft/m.

3.3 CONTROL SURFACE GEOMETRY

3.3.1 Select Geometry Type

After the missile geometry has been defined, the next step is to add the control surfaces. If no control surfaces will be examined, then the 'No Control Surfaces' parameter must be selected in the grid parameter section. Select the desired type of control surface from the drop down menu and click on 'Add Control Surface'.

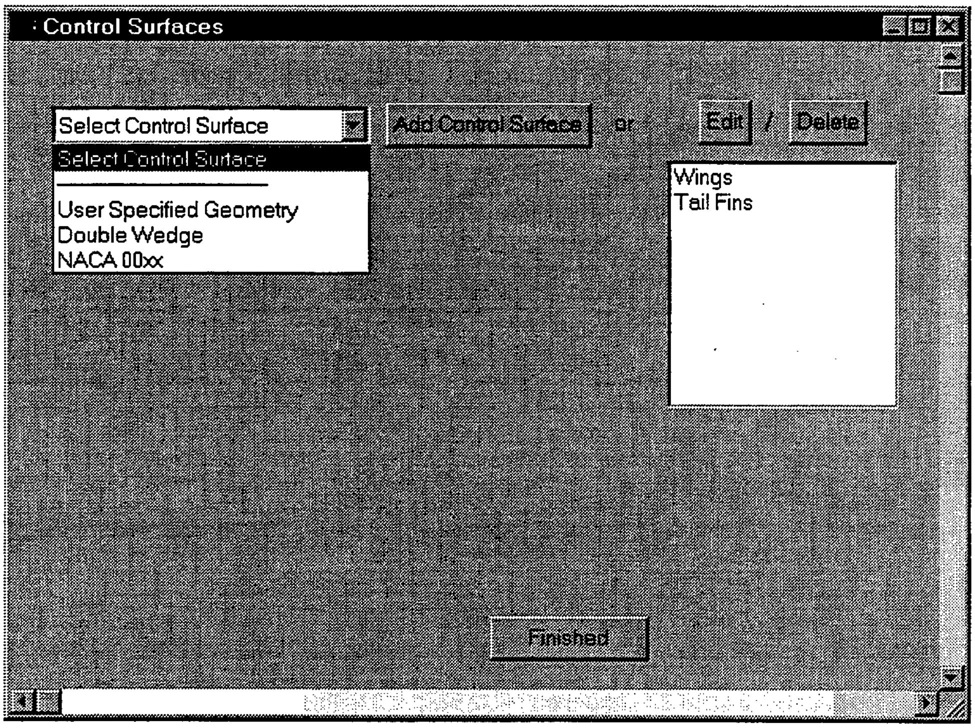
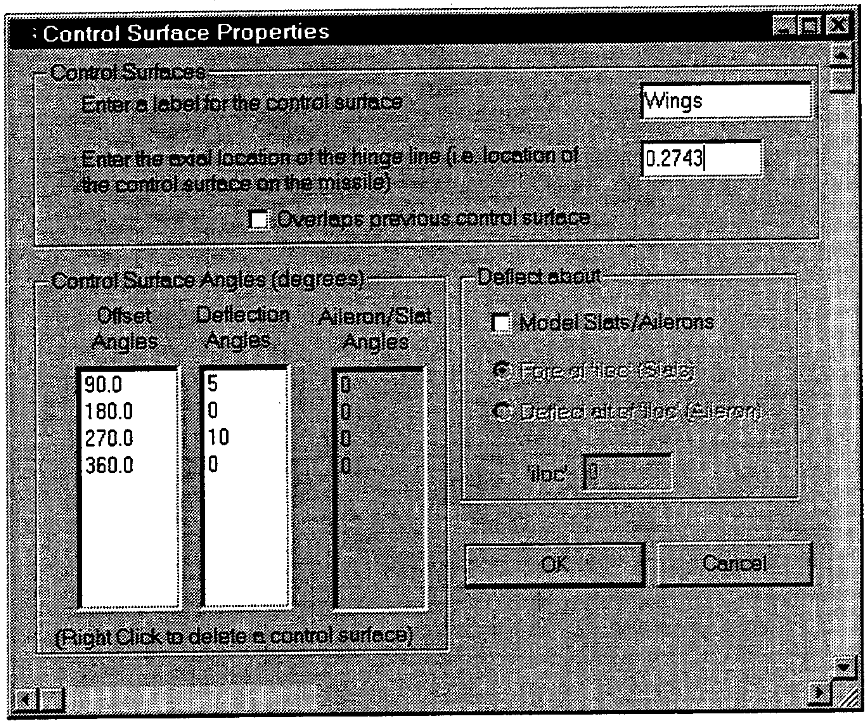


FIGURE 12. CONTROL SURFACE GEOMETRY SELECTION

It should be noted from the above figure, that there are a variety of possible control surfaces. A sample sketch, a description of the required parameters, and the generating equations are shown below for each of the available control surfaces.

3.3.2 Control Surface Parameters

Before specifying the geometry of the control surfaces, a number of parameters (see below) which describe the location/orientation of the surfaces on the missile must be specified. These parameters, along with a description of each, are shown below.



Control Surfaces

Enter a label for the control surface: Wings

Enter the axial location of the hinge line (i.e. location of the control surface on the missile): 0.2743

☐ Overlaps previous control surface

Control Surface Angles (degrees)

Offset Angles	Deflection Angles	Aileron/Slat Angles
90.0	5	0
180.0	0	0
270.0	10	0
360.0	0	0

(Right Click to delete a control surface)

Deflect about

☐ Model Slats/Ailerons

● Fore of 'iloc' (Slats)

○ Deflect aft of 'iloc' (Aileron)

'iloc': 0

FIGURE 13. CONTROL SURFACE PARAMETERS

- *Label section* - Enter a text label that describes the control surface.
- *Hinge line* - Enter the axial distance to the control surface hinge line measured from the missile nose. The hinge line is the point on the control surface about which deflections are made.
- *Overlap previous control surface* - Select this option when two different types of control surfaces are desired at a given axial location (i.e., different size vertical and horizontal stabilizers). A detailed description of this feature is provided in the Linked Fins section.
- *Offset angles* - The offset angles describe the orientation of the fins. For the '+' configuration the corresponding angles are 90°, 180°, 270°, and 360°. Similarly, for the 'x' configuration the angles are 45°, 135°, 225°, and 315°.

  Note: 1) The Zeus^{++} code does not require any form of symmetry, and therefore, any desired combination of offset and/or deflection angles may be specified.

2) The control surfaces must lie on a zonal boundary. Therefore, the zone orientations are equivalent to the offset angles. In other words, the zone edges will occur at the above-specified offset angles.

- *Deflection angles* - The deflection angles are specified in a clockwise manner. In other words, specifying +10 for both the 90° and 270° fins would pitch the former up and the latter down 10°. If a symmetric deflection is desired, the required inputs are +10 for the 90° and -10 for the 270° fin.

  Note: The Zeus^{++} code does not require any form of symmetry, and therefore, any desired combination of offset and/or deflection angles may be specified.

- *Aileron/Slat Angles* - In addition to deflecting the entire surface about the hinge line (see above), it is also possible to deflect a portion of a control surface. This feature allows for the modeling of either Aileron or Slat type controls. To model either a slat or an aileron, first select on the 'Model Slats/Ailerons' and then enter the appropriate deflections in the 'Ailerons/Slats Angles' dialog box. Finally, enter the 'ilocation' of either the slat termination point or the aileron initiation point. The 'iloc' parameter corresponds to the index of the termination/initiation point along each plane defining the control surface and is explained in the User Defined Geometry section.

  Note: When modeling either flaps or slats, it is not possible to deflect a portion of the trailing/leading edges. The entire edge is deflected about the 'iloc' point by the amount specified in the Aileron/Slat Angles dialog box.

- *Delete a fin* - The number of fins is equal to the maximum number of zones specified in the grid parameter section. However, it is possible to delete a fin by right-mouse clicking on the desired fin. Deletion causes the offset, deflection, and the Aileron/Slat angles to be replaced with '——' to indicate that they have been eliminated. To add the fin back into the calculation, simply right click on it a second time. This feature is useful if a variable number of fins are desired for the different sets of control surfaces (i.e., 2 canards and 4 tail fins).

3.3.3 User Defined Geometry

The most general method of specifying the control surface geometry is the 'User Defined Geometry' option. For this method, the actual geometry points (x, y, z) are prescribed by the user. The generality of this method allows nearly any form of symmetric control surface to be examined. Two example cases will be discussed in the current section, the first of which is a simple modified wedge control surface. The figure shown below represents the upper half of the symmetric control surface and would mate to the missile body along edge '1-2-3-4'.

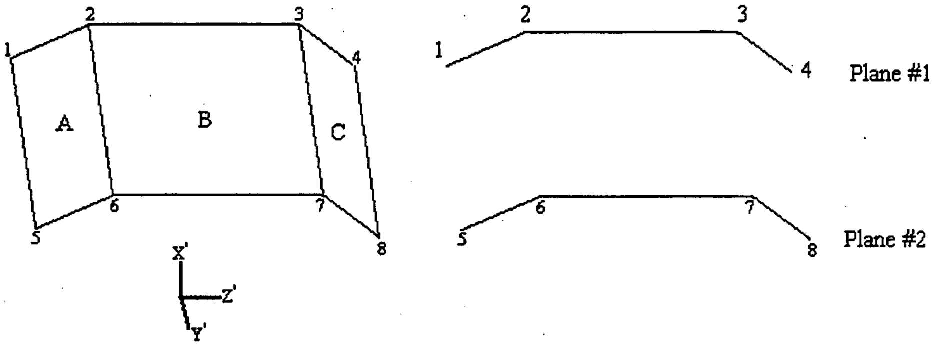


FIGURE 14. SAMPLE 1—USER DEFINED CONTROL SURFACE PARAMETERS

The Zeus^{++} tool requires that the control surfaces be described by a series of planar surfaces (see 'A', 'B', and 'C' above). In other words, 'A', 'B', and 'C' each have a single unit outer normal. This allows the code to calculate unit normals to each section, and hence, to generate a computational domain around the control surface. The geometry is specified by decomposition into a series of planes, all of which must have the same number of node points. For the figure shown above the surfaces 'A', 'B', and 'C' are described by specifying two planes with four points on each plane (Points '1', '2', '3', and '4' define Plane #1, Points '5', '6', '7', and '8' define Plane #2). The first plane (Plane #1) is that which corresponds to the root chord (i.e., lies along the missile surface). Successive planes march outwards until the final plane (Plane #2) is reach at the tip chord.

Note: 1) The coordinate system for entering the control surface points is based on a fin local system.

'x' is the thickness of the control surface.

'y' is the radial distance measured relative to the missile surface. (y=0 corresponds to the surface of the missile).

'z' is the axial distance measured relative to the hinge line.

2) The control surfaces must be symmetric, and therefore, only the upper half of the control surface is specified in the 'User Defined Geometry' section. Because of this symmetry requirement, the leading and trailing edges of the fin must have zero thickness (x=0). This requires that x be zero for the first and last points on each plane describing the control surface.

3) Each plane must contain the same number of node points (four for the above example).

The above example can be used to model either slats or flaps. Note that four points are used to define each of the planes (Plane #1 and Plane #2). To model slats, set the 'iloc' parameter to '2', select the 'Fore of iloc (Slats)' dialog, and specify the slat deflection angle in the Control Surface Parameters section. This causes all points ahead of 'iloc' (surface 'A') to be deflected by the specified slat angle. Similarly, to model flaps, set 'iloc' equal to 3 and select the 'Aft of iloc (Flaps)' dialog to deflect all points aft of the third planar point (i.e., deflect surface 'C').

The second example represents a more complex control surface where three planes must be considered to properly define the geometry.

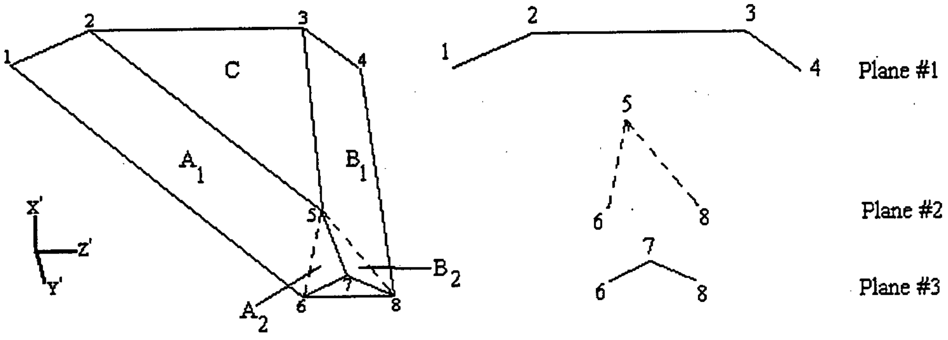
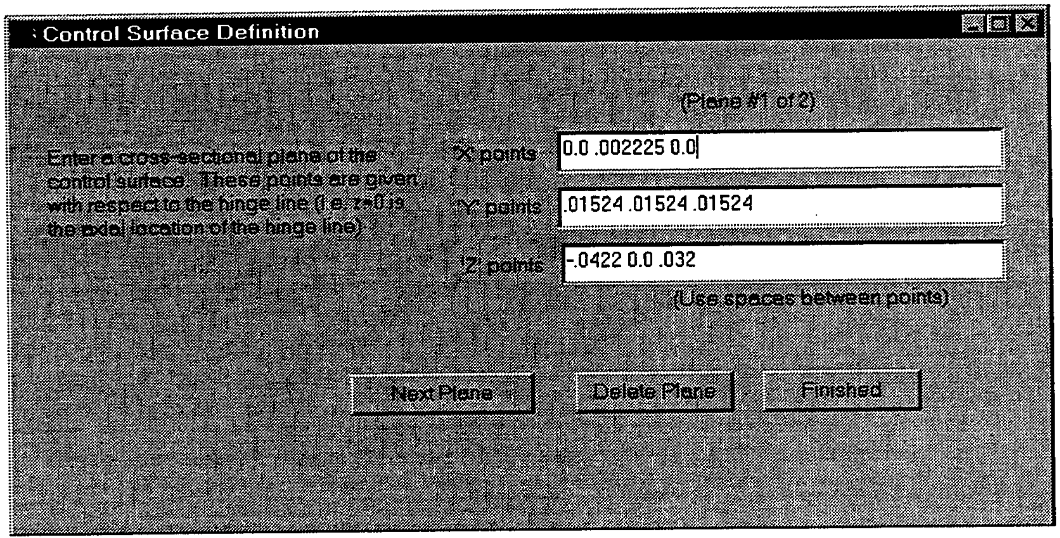


FIGURE 15. SAMPLE 2—USER DEFINED CONTROL SURFACE PARAMETERS

From the above figure it is seen that the surfaces 'A_1' and 'A_2' are not equivalent planar surfaces (i.e., do not have the same unit outer normal). A similar observation is made of the trailing edge of the fin (surfaces 'B_1' and 'B_2'). Therefore, the above surface cannot be described using only Plane #1 and Plane #3 shown above. Plane #2 must be included to properly describe the control surface.

Note: Each plane must contain the same number of nodal points. Plane #1 contains four nodal points; therefore, Planes #2 and #3 must also contain four points. To meet this constraint, simply enter the center point for planes '2' and '3' twice (i.e., duplicate points '5' and '7').

Entry of the 'User Defined Geometry' is performed in the following dialog box. First, enter all of the 'x' points for a given plane, then all of the 'y', and finally, all of the 'z' points. Click the 'Next Plane' button to continue the process until all planes have been entered. Click on 'Finished' after entering all of the geometry points.



(Plane #1 of 2)

Enter a cross-sectional plane of the control surface. These points are given with respect to the hinge line (i.e. z=0 is the axial location of the hinge line)

'X' points: 0.0 .002225 0.0

'Y' points: .01524 .01524 .01524

'Z' points: -.0422 0.0 .032

(Use spaces between points)

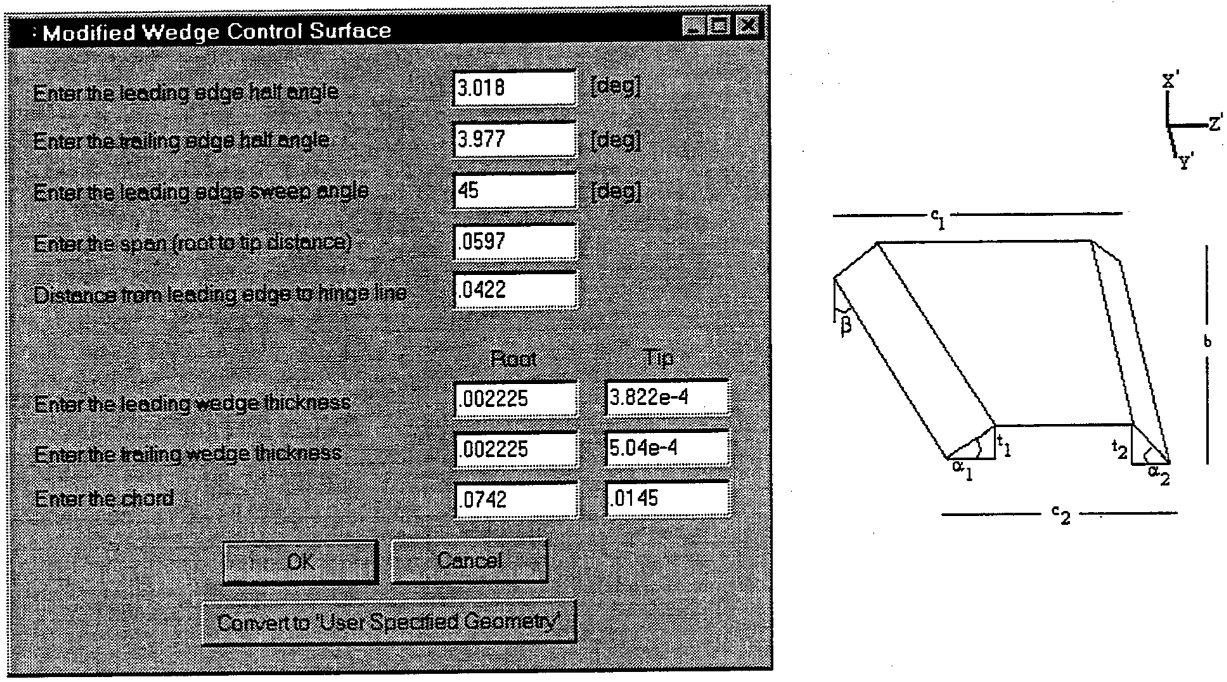
FIGURE 16. USER DEFINED CONTROL SURFACE INPUTS

- *'x', 'y', 'z' points* - Local fin coordinates used for specifying the control surface geometry. 'x' is the fin thickness, 'y' is the radial distance measured from the missile surface, and 'z' is the axial distance measured relative to the hinge line.

- *Planes* - Displays the current plane as well as the maximum plane number.

3.3.4 Modified Wedge

If the control surface is a modified wedge, this section can be used in place of the'User Defined Geometry' to simplify the geometric inputs required by the user.



Modified Wedge Control Surface

Enter the leading edge half angle 3.018 [deg]

Enter the trailing edge half angle 3.977 [deg]

Enter the leading edge sweep angle 45 [deg]

Enter the span (root to tip distance) .0597

Distance from leading edge to hinge line .0422

	Root	Tip
Enter the leading wedge thickness	.002225	3.822e-4
Enter the trailing wedge thickness	.002225	5.04e-4
Enter the chord	.0742	.0145

FIGURE 17. MODIFIED WEDGE CONTROL SURFACE PARAMETERS

- *Half Angles* - Enter both the leading and trailing edge half angles (a_1 and a_2, respectively).

- *Sweep Angle* - Enter the leading edge sweep angle (b)

- *Span* - Enter the control surface span (b)

- *Distance to hinge line* - Enter the distance (on the root chord) from the leading edge to the hinge line.

- *Wedge thickness* - Enter the leading/trailing edge wedge thickness at both the root and the tip of the control surface (t_1 and t_2, respectively).

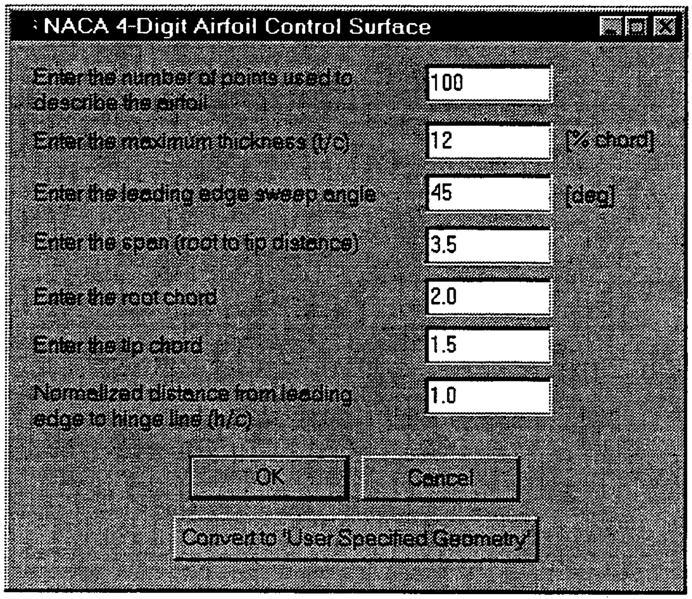
- *Chord* - Enter the root and tip chords (c_1 and c_2, respectively)

- *Convert* - When entering a control surface, it is often desirable to make modifications to the types of control surfaces allowed in Zeus^{++}. If the control surface under consideration is similar to a modified wedge then start by entering all of the aforementioned parameters, then press the 'Convert to User Specified Geometry' button to convert the wedge parameters to 'x', 'y', 'z' coordinates.

It is then possible to modify the individual points to obtain the exact control surface desired.

3.3.5 NACA00xx

This section is used to describe a symmetric NACA00xx airfoil. Note that due to the bluntness of the leading edge, difficulty will most likely be encountered if this fin is evaluated in the Euler solver. It can, however, be used to generate a Navier-Stokes grid.


NACA 4-Digit Airfoil Control Surface

Enter the number of points used to describe the airfoil: 100

Enter the maximum thickness (t/c): 12 [% chord]

Enter the leading edge sweep angle: 45 [deg]

Enter the span (root to tip distance): 3.5

Enter the root chord: 2.0

Enter the tip chord: 1.5

Normalized distance from leading edge to hinge line (h/c): 1.0

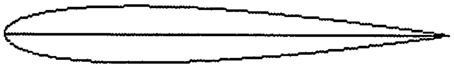

NACA 0012 Symmetric Airfoil

FIGURE 18. NACA 4-DIGIT AIRFOIL PARAMETERS

- *Number of points* - Enter the number of points used to describe the airfoil geometry. Note that increasing this number will not significantly increase the computational time. It is simply the number of points used to draw the two-dimensional airfoil curve.

- *Thickness ratio* - Enter the thickness ratio as a percentage of the chord. (i.e., 12% refers to a thickness of 0.12*chord)

- *Sweep Angle* - Enter the leading edge sweep angle in degrees.

- *Span* - Enter the span (i.e., distance from root to tip).

- *Root/tip chord* - Enter both the root and tip chords.

- *Distance to hinge line* - Enter the normalized distance (h/c) from the leading edge to the hinge line (0-1) along the root chord.

- *Convert* - When entering a control surface, it is often desirable to make modifications to the types of control surfaces allowed in Zeus^{++}. If the control surface under consideration is similar to a NACA 4-digit airfoil then start by entering all of the aforementioned parameters, then press the 'Convert to User Specified Geometry' button to convert the fin parameters to a set of 'x', 'y', 'z' coordinates. It is then possible to modify the node points to obtain the exact control surface desired.

The following is the equation used to calculate the symmetric NACA00xx airfoils (where xx refers to the maximum thickness ratio):

$$\bar{r} = t\left(a_0\bar{z}^{0.5} - a_1\bar{z} - a_2\bar{z}^2 + a_3\bar{z}^3 - a_4\bar{z}^4\right)$$

$$a_0 = 1.4845, \quad a_1 = 0.6300$$

$$a_2 = 1.7580, \quad a_3 = 1.4215$$

$$a_4 = 0.5075$$

$$\bar{r} = \frac{r}{c}, \quad \bar{z} = \frac{z}{c}$$

'r' is the local radius, 'z' is the axial distance, 'c' is the chord, and 't' is the airfoil thickness ratio (thickness/chord).

3.3.6 Linked Fins

It is often desirable to consider different types of control surfaces at a given axial location (i.e., different size vertical and horizontal stabilizers). Zeus^{++} implements this feature by linking two sets of control surfaces together, as described by the following example.

1. Define the first control surface. For example purposes create a 4 finned configuration oriented in the '+' configuration (i.e., fins at 90°, 180°, 270°, and 360°).
2. Delete the 90° and 270° fins on the first control surface.
3. Define a second control surface.
4. Delete the opposing fins on the second control surface (i.e., the 180° and 360°).
5. Select the “Overlap Previous Control Surface” dialog button for the second set of control surfaces.

3.4 GRID GENERATION

Menu Options

In this section, two separate computational domains are examined: the Euler grid and the Navier-Stokes grid. The Euler grid is the one used by the Zeus^{++} tool (see notes below) for the inviscid and boundary layer solvers. The Navier-Stokes grid is generated, along with the appropriate boundary condition files; however, this grid is not used by the Zeus^{++} tool in the flowfield analysis. Rather, this grid (and bc file) is created for use in a full Navier-Stokes solver. A number of menu options are available and are explained below. The first deals with the inviscid Euler grid and the remaining three pertain to the Navier-Stokes grid.

Euler Grid	Generate and view the Euler grid
Generate N-S Grid	Generate the Navier-Stokes grid and boundary condition data files 'filename\RunGrid\GASPGrid.p3da' & 'GASPBC.inp'
Normal Spacing	View the radial spacing (at the first point off the wall) along the missile
View N-S Grid	Plot the Navier-Stokes grid.

3.4.1 Euler Grid

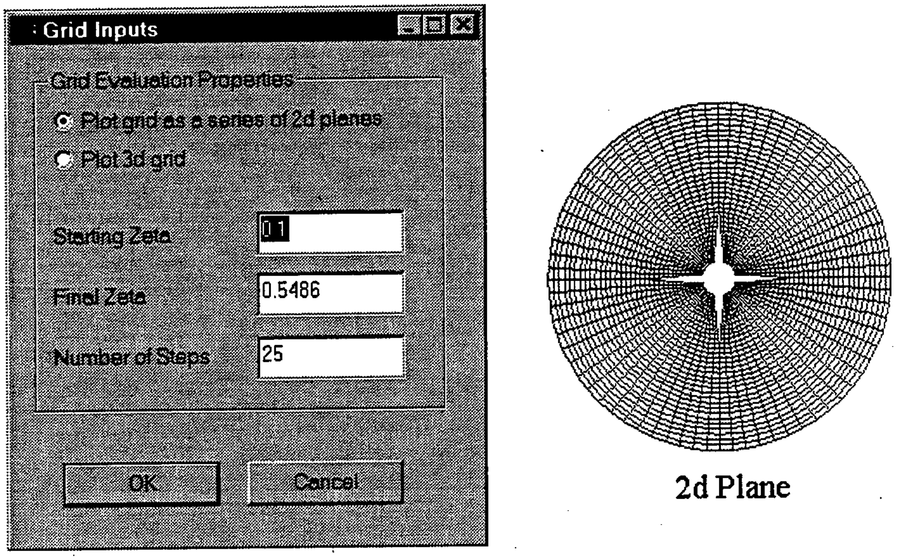
3.4.1.1 View Grid (Optional). Before running the inviscid code, it is often desirable to examine the computational domain to ensure that the relevant geometric features are spatially resolved and to ensure that the control surfaces are input correctly. Two methods of viewing the computational domain are available and are shown in the dialog box below. In the first, the grid is viewed as a series of two-dimensional planes (see sample grid below). The second method displays the Euler grid in three-dimensional block format.

When using the '2d planes' option, it is possible to generate a movie of the grid that steps axially along the missile. To generate the movie file from the two-dimensional grid, run the Tecplot macro 'Playback\2dGrid_Movie.mcr'. The movie file is written to the file 'Playback\Zeus_Grid2d.rm' and can be replayed using the Tecplot Framer program.

> Note: 1) The axial stepsize taken during the inviscid computation is controlled by the Courant-Friederichs-Lewy (CFL) stability condition specified in the integration parameter section. This is calculated at run-time, and therefore, the axial steps taken during the grid generation are not equivalent to those taken during the actual solution. For the purpose of evaluating the computational domain, equal axial spacing is taken along the missile.
>
> 2) There are two possible methods for generating the outer boundary of the computational domain. In the first, it is specified in terms of body radii in the grid parameter section. In the second, the outer boundary tracks the conical shock as the solution proceeds downstream. For the latter method, the conical shock location, and thus the outer boundary is not

known until runtime, and therefore, for the purposes of grid generation, the first method of calculating the outer boundary will be employed.

3) Viewing the Euler grid is not required before executing the inviscid solution. The actual grid used during the inviscid computation is calculated at runtime and can be viewed using the post-processing features of the Zeus^{++} code.



● Plot grid as a series of 2d planes
○ Plot 3d grid

Starting Zeta 0.1

Final Zeta 0.5486

Number of Steps 25

2d Plane

FIGURE 19. EULER GRID GENERATION PARAMETERS

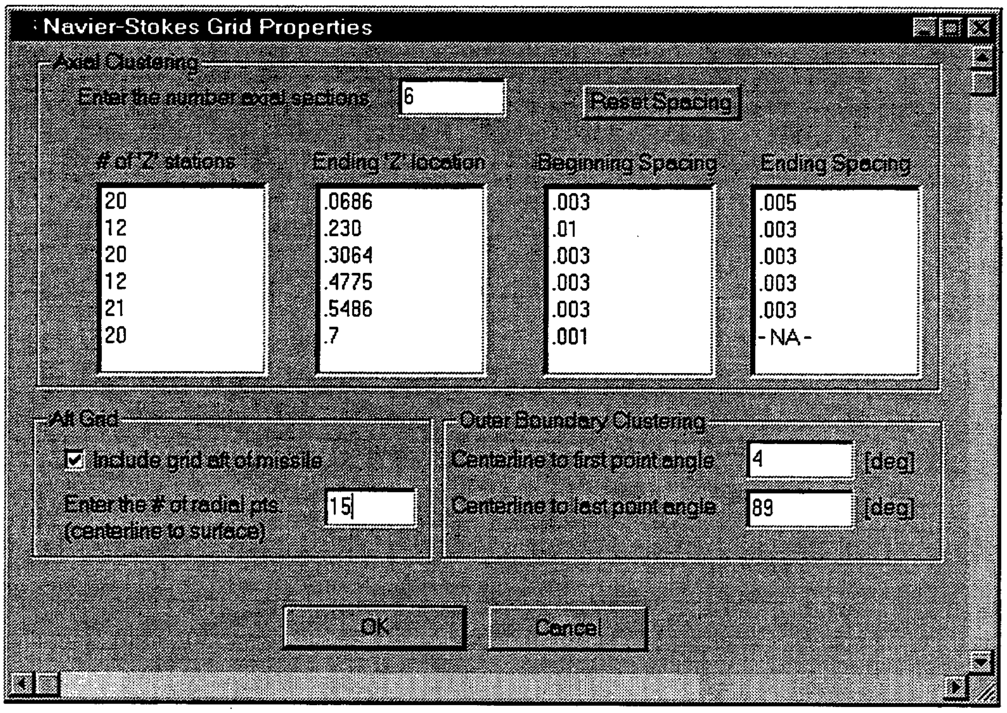
- *Plot type* - Generate the grid as either a series of two-dimensional planes (i.e., each plane is written as a separate Tecplot[9] zone), or a one three-dimensional grid (i.e., block format).

- *Starting/final zeta* - Enter the starting and final axial locations for the grid generation. Any subsection can be viewed using the grid generator, and therefore, the zeta values do not have to correspond to the starting/final axial locations of the missile.

- *Number of steps* - Enter the number of axial steps to be taken. The axial stepsize is assumed constant for the purposes of grid generation and is calculated linearly using the starting and final zeta values specified above.

3.4.2 Navier-Stokes Grid

3.4.2.1 Generate Grid. The Zeus^{++} tool can be used to generate solutions for missile type geometries using both an Euler and a boundary layer solver. However, it is often desirable to perform a more in-depth analysis using a full Navier-Stokes solver. For these cases, the Zeus^{++} tool serves as a grid generator, allowing the user to create complex three-dimensional grids in a matter of minutes. Currently, the tool is setup to generate CFD grids in Plot3d format ('filename\RunGrid\GASPGrid.p3da') along with the appropriate zonal boundary conditions ('filename\RunGrid\GASPBC.inp') for the GASP[1] flow solver. The boundary condition file specifies each point on a zone edge as either a solid surface (a fin) or a zonal boundary.



Axial Clustering

Enter the number axial sections: 6

# of 'Z' stations	Ending 'Z' location	Beginning Spacing	Ending Spacing
20	.0686	.003	.005
12	.230	.01	.003
20	.3064	.003	.003
12	.4775	.003	.003
21	.5486	.003	.003
20	.7	.001	- NA -

Aft Grid

☑ Include grid aft of missile

Enter the # of radial pts. (centerline to surface): 15

Outer Boundary Clustering

Centerline to first point angle: 4 [deg]

Centerline to last point angle: 89 [deg]

FIGURE 20. NAVIER-STOKES GRID GENERATION PARAMETERS

- *Axial clustering* - In this section the axial clustering for the Navier-Stokes grid is specified. The radial and circumferential values are declared in the grid parameter section.

Note: 1) Zeus^{++} uses a Newton method to calculate the axial clustering parameters (i.e., geometric stretching factors). To prevent a divergent solution, a valid initial spacing must be provided. This is accomplished by first specifying the number of axial sections along the missile. Next, enter the number of points and the ending axial location for each section. Finally, click on the 'Reset Spacing' button to calculate the appropriate parameters for an equally spaced grid. Once a valid grid has been obtained, the beginning/ending spacing can be adjusted incrementally to obtain the desired axial clustering.

2) The grid and boundary condition output files produced during the grid generation process are often times very large, and therefore, may take some time to generate. Also, the process of determining the appropriate axial clustering is an iterative procedure, and is independent of the circumferential layout of the grid. Therefore, to minimize the amount of calculation required per iteration, set the number of circumferential points to unity in the Navier-Stokes grid section. After the desired axial clustering is obtained, reset the number of circumferential points to the desired value, and regenerate the final Navier-Stokes grid.

- *Number of axial stations* - Specify the number of axial sections in which clustering is desired. For the sample grid shown below the six sections designated are distinguished by the vertical tic marks. The division shown in the example allows for axial clustering at the beginning/end of the nose section, leading/trailing edges of both sets of fins, as well as clustering at the aft end of the missile.
- *Number of 'z' stations* - The number of axial points per section.
- *Ending 'z' location* - Ending axial location for each section [ft/m].
- *Reset spacing* - Reset the beginning/ending axial spacing in each section to obtain equal spacing in all sections.
- *Beginning spacing* - Beginning axial spacing (Δz) for each section [ft/m].
- *Ending spacing* - Ending axial spacing (Δz) for each section [ft/m].
- *Aft grid* - If a grid aft of the missile is desired (subsonic flow) then this checkbox must be selected.
- *Number of radial points* - This refers to the points which lie radially between the missile centerline and the missile surface directly behind the missile (see yellow grid section or area designated by the horizontal tic marks). The radial spacing at the first point is taken to be the same as that on the surface (at the aft end) of the missile. Zeus^{++} calculates the geometric stretching parameter that places the final radial point along the missile centerline. If the error message 'Unattainable radial clustering' is displayed, try reducing the number of radial points in the aft grid section.
- *Outer boundary clustering* - Specify the outer boundary clustering around the nose section of the missile.
- *First point angle* - The angle between the missile centerline and the first nose point [degrees]. See angle 'a' below. *The angle must be greater than 0 degrees.*
- *Last point angle* - The angle between the missile centerline and the final nose point [degrees]. See angle 'b' below. *The angle must be less than 90 degrees.*

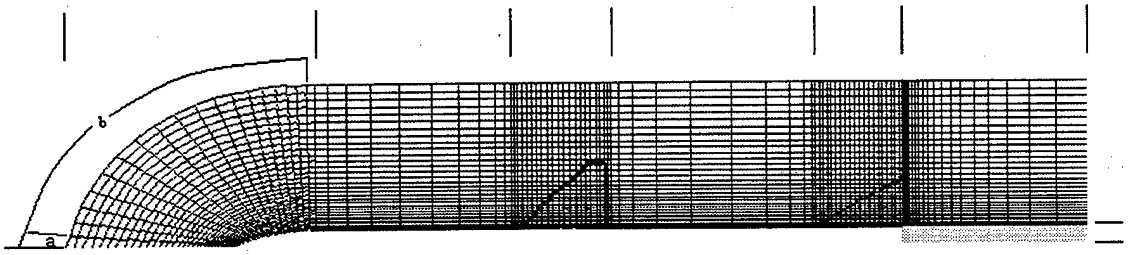
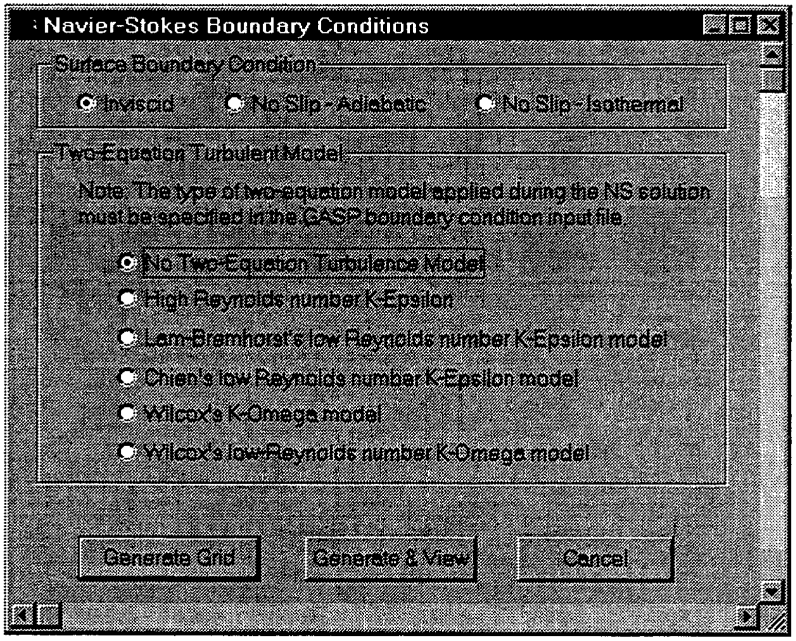


FIGURE 21. SAMPLE CROSS-SECTION OF NAVIER-STOKES GRID

The final step in generating a Navier-Stokes grid is to specify the surface boundary conditions (see figure below).



Navier-Stokes Boundary Conditions

Surface Boundary Condition

● Inviscid ○ No Slip - Adiabatic ○ No Slip - Isothermal

Two-Equation Turbulent Model

Note: The type of two-equation model applied during the NS solution must be specified in the GASP boundary condition input file.

● No Two-Equation Turbulence Model
○ High Reynolds number K-Epsilon
○ Lam-Bremhorst's low Reynolds number K-Epsilon model
○ Chien's low Reynolds number K-Epsilon model
○ Wilcox's K-Omega model
○ Wilcox's low-Reynolds number K-Omega model

FIGURE 22. NAVIER-STOKES BOUNDARY CONDITION PARAMETERS

- *Surface boundary condition* - Specify the type of boundary condition to be employed when a solid surface is encountered. Currently, the Zeus^{++} tool generates a boundary condition file for the GASP flow solver, which specifies the boundary condition along the zone edges (i.e., solid surface or zonal boundary).

- *Two-Equation Model* - The type of two-equation turbulence model applied during the Navier-Stokes solution must be specified in the GASP boundary condition input file.

3.4.2.2 View Normal Spacing. A prerequisite for accurately computing a Navier-Stokes solution is the spatial resolution of the boundary layer, and therefore, the computational grid must be appropriately refined near the surface. The boundary layer thickness cannot be determined a priori, however, an approximate value can be obtained using flat plate assumptions. Using this estimate for the boundary layer thickness, the radial clustering parameter can be increased/decreased until an appropriate value for the normal spacing to the first point off the surface is obtained. A typical value for the required normal spacing is 1/20th to 1/30th of the approximate boundary layer thickness (i.e., approximately 20-30 points in the boundary layer).

3.4.3.3 View Grid. This option plots the Navier-Stokes grid in three-dimensional block format using the Tecplot[9] plotting package. A two-dimensional slice of the three-dimensional grid can be obtained by executing the Tecplot macro 'Playback\GaspGrid.mcr'. This is the view displayed in the Generate Gasp Grid section, and is useful for evaluating the axial clustering parameters as well as the spatial resolution of the control surfaces.

> Note: The control surfaces are represented by the contour plot variable 'onfin'. Unity and zero represent a solid surface and a zonal boundary, respectively. To evaluate the spatial resolution of the control surfaces, examine a contour plot of the 'onfin' parameter.

CHAPTER 4: INPUT PARAMETERS

OVERVIEW

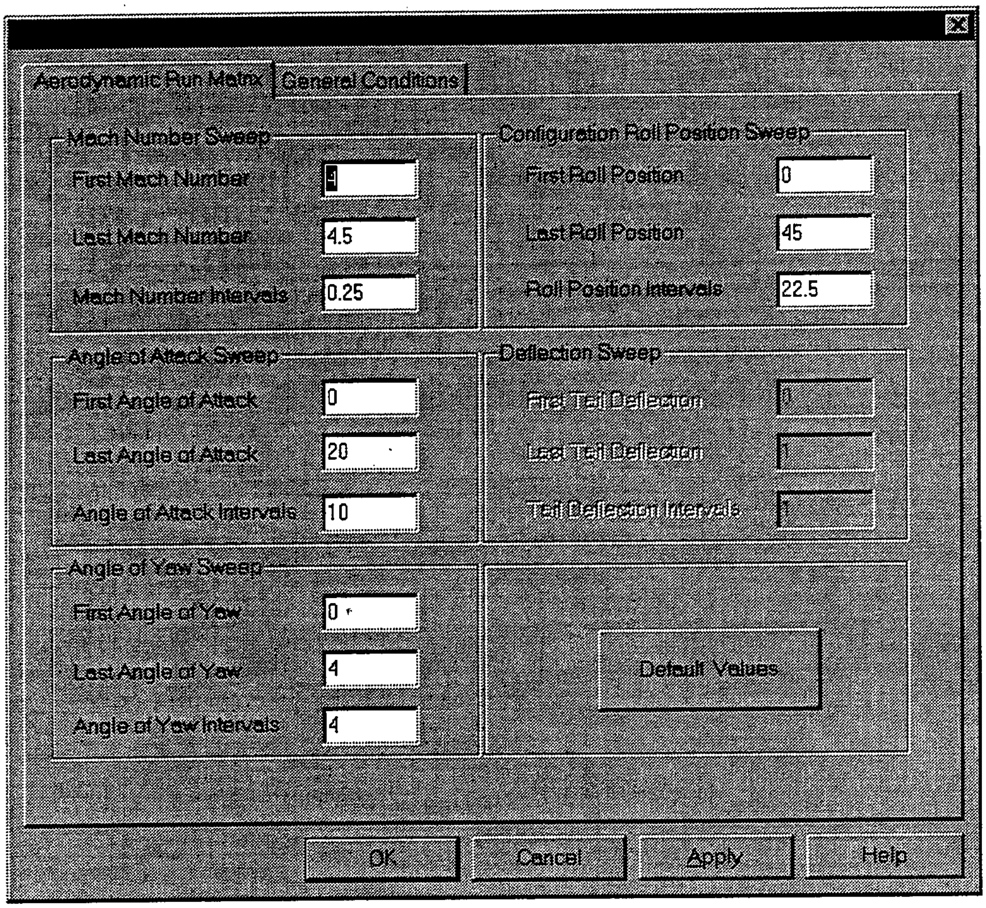
A variety of parameters are required before execution of the Euler or boundary layer solvers can proceed. The runtime parameters are specified in the 'Inputs' section from the main menu. The parameters are separated into four basic categories as shown below.

1. Aerodynamic Data
    - <u>Run Matrix</u>
    - <u>General Data</u>
2. <u>Integration Controls</u>
3. <u>Separation Parameters</u>
4. Boundary Layer Controls
    - <u>Execution Parameters</u>
    - <u>Modeling Parameters</u>
    - <u>Output Parameters</u>
5. <u>Output Controls</u>

4.1 AERODYNAMIC PARAMETERS

4.1.1 <u>Run Matrix Parameters</u>

The Aerodynamic run matrix is where the Mach numbers and missile orientations are specified. This is accomplished by entering an initial, final, and incremental value for Mach number, angle of attack, angle of yaw, and roll angle (see figure below). Zeus^{++} then calculates an Aerodynamic run matrix consisting of all possible cases.



Mach Number Sweep
First Mach Number: 4
Last Mach Number: 4.5
Mach Number Intervals: 0.25

Configuration Roll Position Sweep
First Roll Position: 0
Last Roll Position: 45
Roll Position Intervals: 22.5

Angle of Attack Sweep
First Angle of Attack: 0
Last Angle of Attack: 20
Angle of Attack Intervals: 10

Deflection Sweep
First Tail Deflection: 0
Last Tail Deflection: 1
Tail Deflection Intervals: 1

Angle of Yaw Sweep
First Angle of Yaw: 0
Last Angle of Yaw: 4
Angle of Yaw Intervals: 4

FIGURE 23. AERODYNAMIC SWEEP PARAMETERS

- *Mach number sweep* - Enter the starting, final, and incremental Mach number. For the sample shown above, a Mach number of 4.0, 4.25, and 4.50 will be considered.

- *Angle of attack sweep* - Enter the starting, final, and incremental angle of attack.

- *Angle of yaw sweep* - Enter the starting, final, and incremental yaw angle.

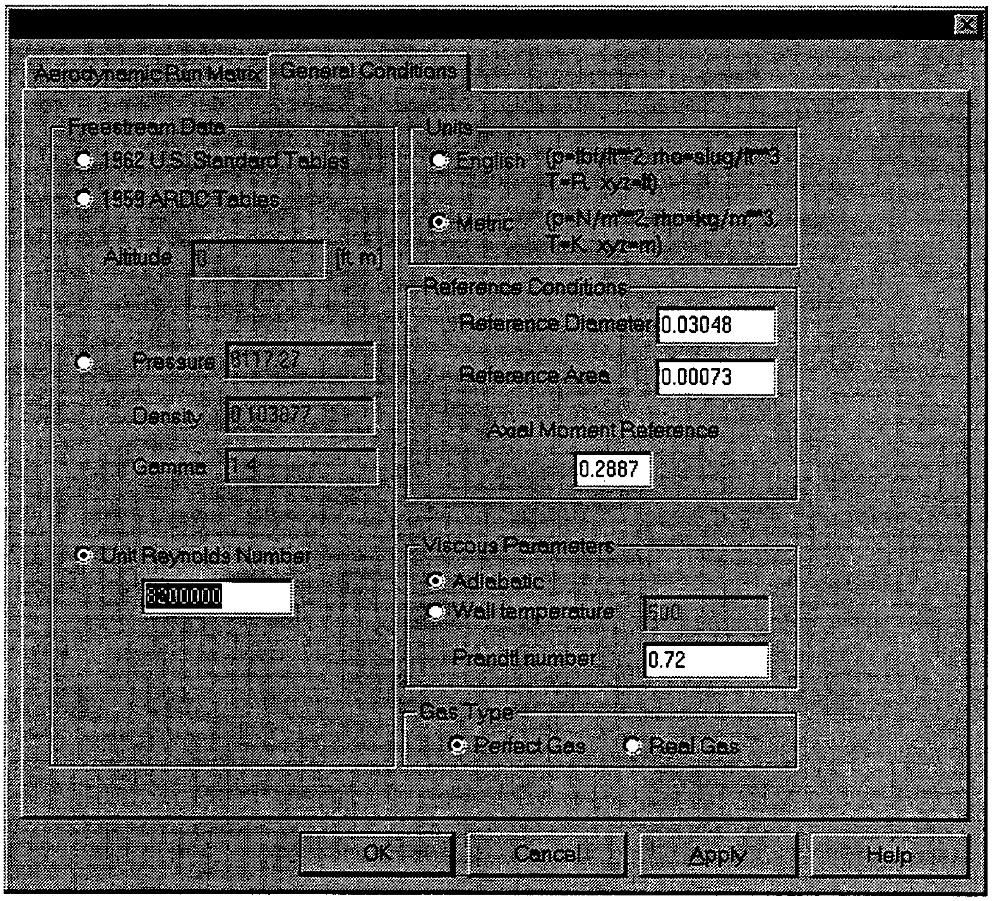
- *Roll angle sweep* - Enter the starting, final, and incremental roll angle.

- *Deflection sweep* - Running sweeps on control surface deflections has not been implemented in the current version.

  Note: For each case (i.e., every Mach number, angle of attack, angle of yaw, and roll angle combination) a subdirectory will be created under the main directory and will be labeled Run0001-Runxxxx. In addition, a file called RunMatrix.txt is generated which maps the case number to the aerodynamic parameters entered above.

4.1.2 General Aerodynamic Parameters

A number of general aerodynamic parameters must be specified before executing either the Euler or boundary layer solvers. These parameters are shown in the following figure and described in detail below.



○ 1962 U.S. Standard Tables
○ 1959 ARDC Tables
Altitude: 0 [ft, m]
○ Pressure: 8117.27
Density: 0.103877
Gamma: 1.4
● Unit Reynolds Number: 8200000

○ English (p=lbf/ft**2, rho=slug/ft**3, T=R, xyz=ft)
● Metric (p=N/m**2, rho=kg/m**3, T=K, xyz=m)

Reference Diameter: 0.03048
Reference Area: 0.00073
Axial Moment Reference: 0.2887

● Adiabatic
○ Wall temperature: 500
Prandtl number: 0.72

● Perfect Gas ○ Real Gas

FIGURE 24. GENERAL AERODYNAMIC INPUT PARAMETERS

- *Free-stream data* - The required free-stream data includes the pressure, density, and ratio of specific heats (g). There are three methods of specifying these parameters.

    1) Select either of the standard atmospheric tables (1962 U.S. Standard Tables or 1959 AFDC Tables) and then enter an altitude in the provided dialog box. The pressure and density will be calculated from the standard table data and the ratio of specific heats is assumed that of standard air (1.4).
    2) Specify the pressure, density, and ratio of specific heats directly by clicking on the radio button next to these parameters and entering the desired values.

3) Specify a unit Reynolds number and let Zeus^{++} calculate the corresponding pressure and density. Gamma, the ratio of specific heats, is taken to be that of standard air (1.4).

$$\mathrm{Re}_L = \frac{\rho_\infty U_\infty}{\mu_\infty}$$

- *Units* - Specify either English or metric units.

  Note: All of the parameters entered must be in base units (i.e., meters, not centimeters; feet, not inches, ...etc.). Incorrect results will be obtained if you enter all of your lengths in centimeters and then try to compensate by entering the reference quantities in centimeters. Although entering non-standard units will not effect the inviscid solution, it will produce incorrect results for the viscous solution due to the Reynolds number effect.

- *Reference conditions* - Enter the reference diameter (typically the base diameter) and the reference area (typically the area calculated using the base of the missile). Also, specify the axial location about which the pitching and yawing moments are calculated (The rolling moment is always taken about the missile centerline).

  Note: A nose down pitching moment is negative.

- *Viscous parameters* - If the viscous boundary layer code is to be considered then the wall boundary condition for the heat flux and the Prandtl number must be specified. For the adiabatic wall, calculating an appropriate wall temperature using the recovery factor methodology imposes a zero heat flux boundary condition.

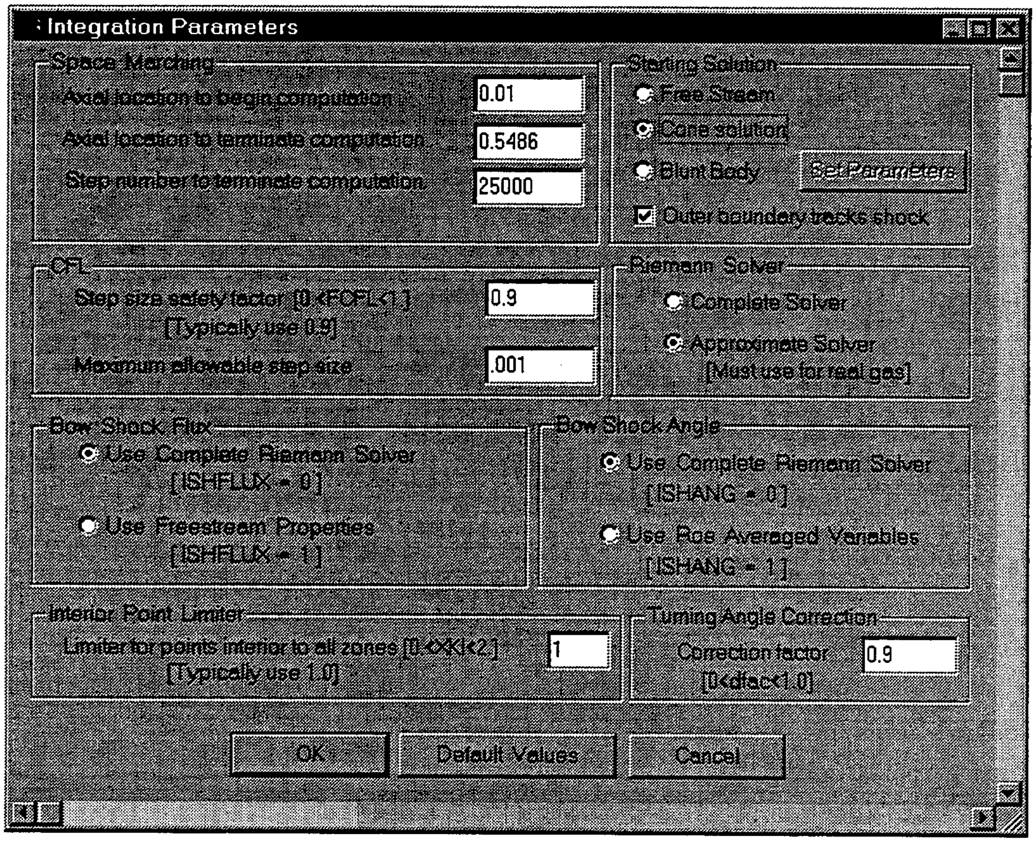
- *Gas type* - Select either a perfect gas or a real gas solution.

  Note: If 'real gas' is selected then the approximate Riemann solver must be selected in the integration controls section.

4.2 INTEGRATION CONTROLS

4.2.1 Parameter Description

A number of user specified integration parameters must be specified in the following section. These include axial limits for integration, starting solution, stability criterion, limiters, and flux calculation parameters.



Integration Parameters

Space Marching
Axial location to begin computation: 0.01
Axial location to terminate computation: 0.5486
Step number to terminate computation: 25000

Starting Solution
○ Free Stream
● Cone solution
○ Blunt Body
☑ Outer boundary tracks shock

CFL
Step size safety factor [0<FCFL<1.]: 0.9
[Typically use 0.9]
Maximum allowable step size: .001

Riemann Solver
○ Complete Solver
● Approximate Solver
[Must use for real gas]

Bow Shock Flux
● Use Complete Riemann Solver
[ISHFLUX = 0]
○ Use Freestream Properties
[ISHFLUX = 1]

Bow Shock Angle
● Use Complete Riemann Solver
[ISHANG = 0]
○ Use Roe Averaged Variables
[ISHANG = 1]

Interior Point Limiter
Limiter for points interior to all zones [0.<XKI<2.]: 1
[Typically use 1.0]

Turning Angle Correction
Correction factor 0.9
[0<dfac<1.0]

FIGURE 25. NUMERICAL INTEGRATION CONTROLS

- *Space marching* - Enter the starting and final axial locations for the solution. Also, enter a maximum number of steps to take before the code aborts.

  Note: The starting axial location is dependent on the type of starting solution applied (see below for an explanation of each of the different methods of generating a starting solution).

- *Stability criterion* - Enter a step size safety factor (analogous to a CFL number) between zero and unity. Also, specify a maximum allowable step size to ensure that all relevant features are spatially resolved in the axial direction.

- *Starting solution* - The Euler solver is a marching code, and therefore, a starting solution must be specified on the missile before the integration can proceed. Three different types of starting solutions are available, each of which is discussed below.
  1) Free-stream - A free-stream profile is specified as the starting solution at the axial location entered in the 'space Marching' section. For this type of inflow profile, the outer boundary does not track the conical shock, but rather, is user specified in terms of body radii in the <u>grid parameter</u> section.

  Note: The region upstream of this profile is ignored, and therefore, in order to accurately compute the forces and moments, the integration should begin as close to the nose as is numerically possible.

2) Cone Solution - The cone starting solution applies a one-dimensional cone solver to the region from the nose to the beginning axial location specified in the space marching section. This calculates the shock angle, as well as the flowfield directly aft of the shock, given the local turning angle. The starting profile is then taken to be that generated by the conical solver. In other words, a conical flow solver is applied from the nose tip to the location specified as the 'Axial Location to begin computation'. The Euler solver then begins integration at this location with the profile supplied by the conical flow solver.

Note: a) The conical flow solver is intended to provide the shock angle as well as the flowfield properties directly aft of the shock. It is not intended to solve the flowfield for the entire nose cone. The one-dimensional cone solver cannot calculate the forces/moments, and therefore, if you do not start the Euler solver until the end of the nose section, the forces/moments due to this section will be neglected. In other words, the conical solver should only be applied to a very small region around the nose tip and not to the entire nose section.

b) If the local turning angle becomes excessively large then the one-dimensional cone solver will compute a subsonic Mach number aft of the shockwave. The Zeus Euler solver is a space marching code, and therefore, cannot handle subsonic regions. For this case, a free-stream starting solution must be applied.

3) Blunt body - For a blunt body, the flow will be subsonic in some region surrounding the nose section. If you attempt to begin the Zeus integration too close to the nose, the code will encounter a subsonic region and abort. To prevent this, a blunt body solver must be applied to the entire subsonic region. This solver provides a solution from the beginning of the nose to the value specified in the space marching section labeled 'Axial location to begin computation'. The user must ensure that this value is downstream of the entire subsonic region.

Note: In contrast to the free-stream and conical starting profiles, the blunt body solver calculates the forces and moments in the region between the nose and the starting location for the Euler solver. These forces and moments are summed with the values obtained by the Euler solver to provide the total forces/moments acting on the body. Thus, for this starting procedure, it is acceptable to run the blunt body solver over the entire nose section.

- *Riemann solver* - Specify whether the full Riemann or approximate Riemann solver is applied.

- *Bow shock flux* - Calculate the numerical flux at the bow shock using either the complete Riemann solver or freestream properties.

- *Bow shock angle* - Calculate the bow shock angle using either the complete Riemann solver or the Roe averaged variables.

- *Interior point limiter* - Numerical limiter at all interior points (i.e., not adjacent to a surface). The default value is unity, however, this can be increased/decreased to modify the numerical damping provided by the integration scheme. The applicable range is 2 to 0, where 2 and 0 are the least and most dissipative, respectively. A value of zero reduces the integration scheme to a first-order Gudonov method.

  Note: Decreasing the damping (i.e., increasing the limiter value above) will improve the accuracy of the numerical scheme and will more accurately capture the flowfield discontinuities. Insufficient damping can often lead to numerical oscillations in the flowfield and prevent a stable integration.

- *Turning angle correction* - If a subsonic region is encountered while turning the flow, the Zeus code will apply a limiter known as a turning angle correction. The actual turning angle encountered in the flow is multiplied by the turning angle correction to try to prevent a subsonic region. If this correction does not prevent a subsonic region then the code will abort. Typically, a value of 0.9 (i.e., 90% of the original angle) is applied.

4.3 SEPARATION PARAMETERS

4.3.1 Parameter Description

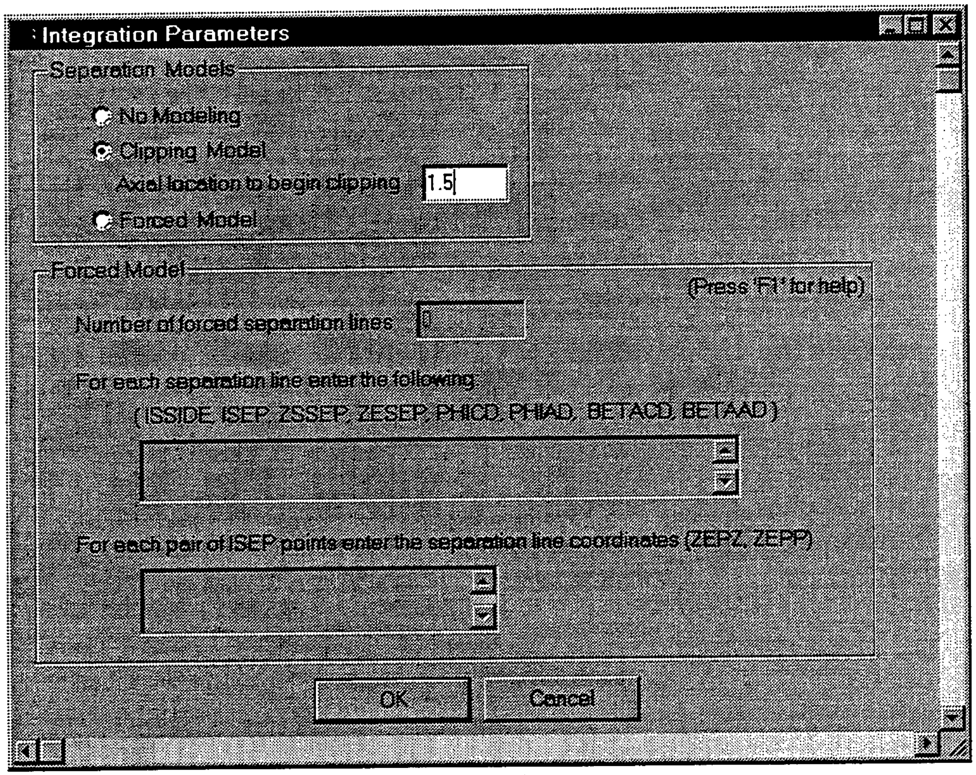
The Zeus code is an inviscid solver, and as such, neglects the effects of viscosity on the flowfield. Since flow separation is a viscous phenomenon, an Euler solver alone cannot accurately predict the effects of a separation zone on the flowfield. Two separation models (clipping and forced) are available in the Zeus Euler solver which attempt to improve the predictive capabilities of the tool near separation zones.

The clipping separation model is described in detail in Ref.11 and operates by decreasing the crossflow velocity on and near the body surface. The crossflow velocity reduction is accomplished by setting an upper limit to the allowable crossflow velocity. If the velocity at any point exceeds this value, it is reduced to this level. Pressure and density are assumed unchanged and the axial component of velocity is increased to give the correct stagnation enthalpy value. Clipping destroys the crossflow shock and produces a large vortex on the leeside of the body that is in qualitative agreement with experiment. Computed pressures on the leeside of the body are in better agreement with experiment. However, clipping tends to increase the windward pressures and often decreases the accuracy of the results in this region.

The forced separation model, originally described in this form by Ref. 12, seeks to simulate separation by altering the velocity direction along a user defined separation line. In each crossflow plane, the separation model is applied to wall cells "s" and "s+1", which are immediately windward and leeward of the separation line, respectively. Pressure and density at these points are defined by:

$$
\begin{aligned}
p_s &= 0.5\left(p'_{s-1} + p'_{s+1}\right) \\
\rho_s &= 0.5\left(\rho'_{s-1} + \rho'_{s+1}\right) \\
p_{s+1} &= 0.5\left(p'_s + p'_{s+2}\right) \\
\rho_{s+1} &= 0.5\left(\rho'_s + \rho'_{s+2}\right)
\end{aligned}
$$

Here $'$ denotes old values. The flow velocities are determined by prescribing a streamline direction and assuming constant stagnation enthalpy. Unfortunately, solutions obtained with the forced separation model are sensitive to mesh size. As the mesh is refined, the pressure beneath the vortex diminishes.



Integration Parameters

Separation Models

○ No Modeling
● Clipping Model
Axial location to begin clipping 1.5
○ Forced Model

Forced Model

(Press 'F1' for help)

Number of forced separation lines 0

For each separation line enter the following:
(ISSIDE, ISEP, ZSSEP, ZESEP, PHICD, PHIAD, BETACD, BETAAD)

For each pair of ISEP points enter the separation line coordinates (ZEPZ, ZEPP)

FIGURE 26. SEPARATION MODELING PARAMETERS

- *Separation Model* - Select the desired separation model.

- *Clipping Separation* - Enter the axial location to begin clipping.

- *Forced Separation* - Enter the number of forced separation lines. For each of the lines, the user must specify the following parameters:

- *ISIDE* - The side of the body on which the separation occurs. Zero if separation occurs between 0° and 180°, unity if separation occurs between 180° and 360°.

- *ISEP* - The number of points used to define the separation line.

- *ZSSEP* - Zeta value at which separation is started.

- *ZESEP* - Zeta value at which separation is terminated.

- *PHICD, PHIAD, BETACD, BETAD* - Flow direction in degrees. Typical values are 20°, 20°, 20°, 5°, respectively.

- *ZEPZ,ZEPP* - A list of ISEP pairs of separation line coordinates where ZEPZ and ZEPP are the z and φ coordinates for each point.

4.4 BOUNDARY LAYER CONTROLS

4.4.1 BL Execution Parameters

The boundary layer code requires, as a boundary condition, the inviscid solution from the Zeus code. Therefore, before executing the boundary layer code, Zeus^{++} must save the inviscid solution at a number of different axial stations. It is both unnecessary and infeasible (due to computational constraints) to save the inviscid flowfield at every axial station, however, an adequate number of axial stations must be considered in order to accurately solve the boundary layer equations. The user must save the inviscid solution often enough to resolve all relevant geometric features axially (i.e., must have several axial stations written out on each control surface).

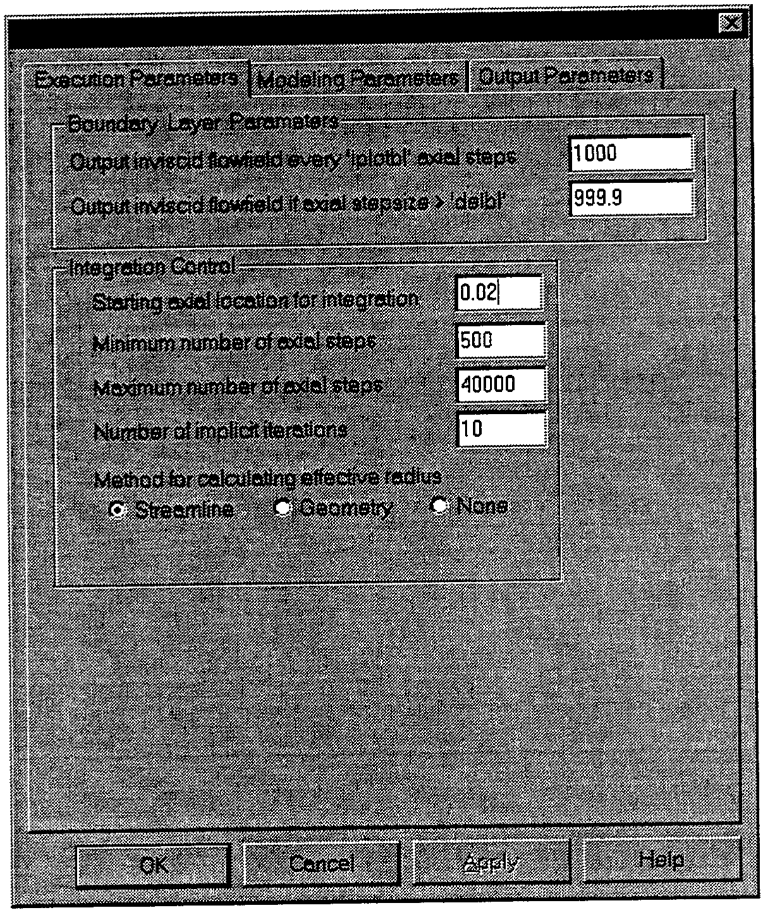
- *Output inviscid flowfield* – Enter the increment and the axial stepsize for saving the inviscid flowfield. The Zeus code will save the inviscid flowfield every incremental axial step as well as every time the axial stepsize exceeds the specified value.

  Note: The Zeus^{++} code must save the inviscid flowfield often enough to axially resolve all relevant geometric features.

- *Starting axial location for integration* – Enter the axial location to begin the boundary layer solver. Note that this must lie downstream of the starting location of the Euler solver as specified in the Integration Parameters section.

- *Minimum number of steps* – Enter the minimum number of axial steps the boundary layer solver will take. The code uses this parameter to calculate a maximum allowable stepsize.

- *Maximum number of steps* – Enter the maximum number of axial steps the boundary layer solver will take before aborting.



Boundary Layer Parameters

Output inviscid flowfield every 'iplotbl' axial steps: 1000

Output inviscid flowfield if axial stepsize > 'delbl': 999.9

Integration Control

Starting axial location for integration: 0.02

Minimum number of axial steps: 500

Maximum number of axial steps: 40000

Number of implicit iterations: 10

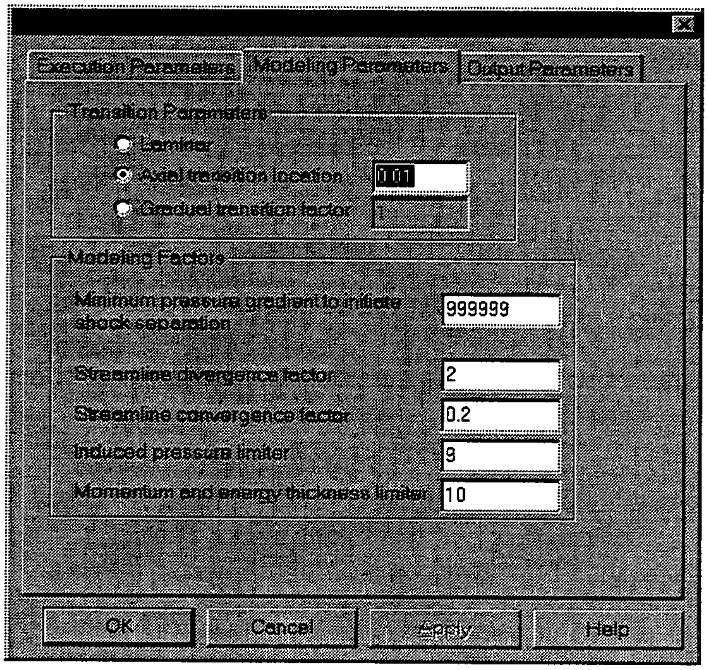
Method for calculating effective radius

● Streamline ○ Geometry ○ None

FIGURE 27. BOUNDARY LAYER EXECUTION PARAMETERS

- *Implicit iterations* – Number of implicit iterations.
- *Effective radius* – This parameter selects the manner in which the effective radius is calculated; the method by which streamline spreading is determined. The "streamline" option uses the distance between successive streamlines, the "geometry" option computes the local surface radius, while "none" sets the radius to unity. Optimal results are obtained with the "streamline" option except on circular bodies at zero incidence. Here the "geometry" option is recommended.

4.4.2 BL Modeling Parameters



Transition Parameters

○ Laminar
● Axial transition location: 0.01
○ Gradual transition factor: 1

Modeling Factors

Minimum pressure gradient to initiate shock separation: 999999
Streamline divergence factor: 2
Streamline convergence factor: 0.2
Induced pressure limiter: 9
Momentum and energy thickness limiter: 10

FIGURE 28. BOUNDARY LAYER MODELING PARAMETERS

- *Transition parameters* - Select one of the three possible transition parameters. If laminar is selected then the flow is forced to remain laminar for the entire calculation. If an 'Axial transition location' is specified, the flow is laminar up to this point and then abruptly transitions to turbulent flow. If the 'Gradual transition factor' is selected, then the transition model is taken from Ref. 6, and the parameter specified is F_t (default of 1.0) from Eq. 23.

- *Pressure gradient for separation* - If the streamwise pressure gradient exceeds this user specified parameter the boundary layer and energy thicknesses are reduced to simulate separation[8].

- *Streamline divergence* - Controls streamline divergence as described in Reference 6 (K_d from Eq. 32). Set to unity to follow inviscid streamlines. Use a value of 2.0 for bodies at incidence. Only applies to edge #1 of each zone.

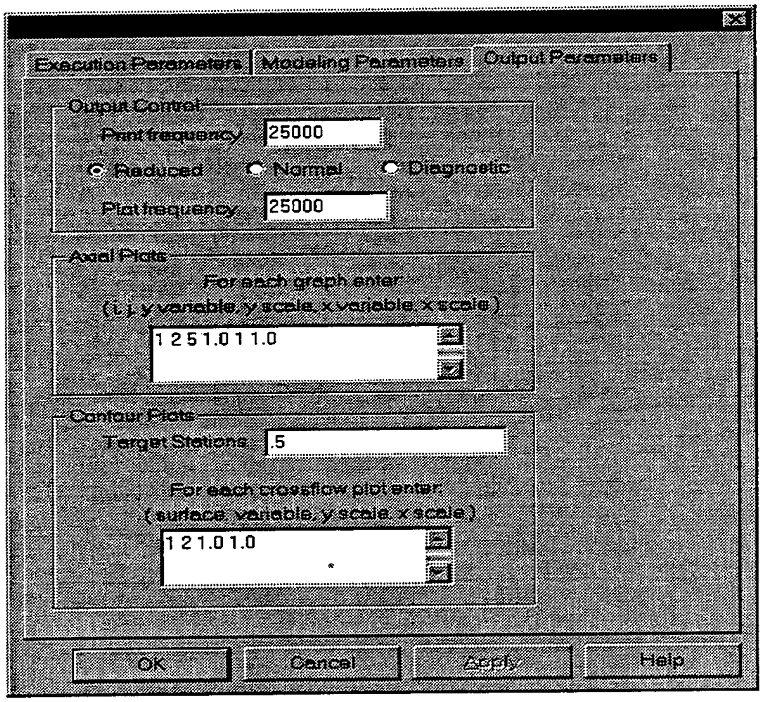
- *Streamline convergence* - Controls streamline convergence as described in Reference 6 (K_c from Eq. 32). Set to unity to follow inviscid streamlines. Use a value of 0.2 for bodies at incidence. Only applies to edge #1 of each zone.

- *Induced pressure limiter* - The boundary layer code attempts to correct for the induced pressure effects caused by the formation of the boundary layer. To prevent large pressure corrections around leading/trailing edges, the induced pressure is limited using the 'Induced pressure limiter' as discussed in Ref. 6 (K_f in section 4.3).

- *Momentum/Energy thickness limiter* - A parameter which limits the momentum/energy thickness in anticipated regions of separation. The limiter is discussed in Ref. 6 (k from Eq. 33).

4.4.3 BL Output Controls

The following dialog box controls the output from the boundary layer solver.



Output Control

Print frequency 25000

● Reduced ○ Normal ○ Diagnostic

Plot frequency 25000

Axial Plots

For each graph enter

(i, j, y variable, y scale, x variable, x scale)

1 2 5 1.0 1 1.0

Contour Plots

Target Stations .5

For each crossflow plot enter:

(surface, variable, y scale, x scale)

1 2 1.0 1.0

FIGURE 29. BOUNDARY LAYER OUTPUT PARAMETERS

- *Print Frequency* - The ASCII output table, 'ZeusBL.out' is generated every 'print frequency' steps. The 'reduced/normal/diagnostic' option controls the amount of data written to the output tape. "Reduced" prints only the most important quantities such as C_f and C_h. "Normal" includes additional boundary layer parameters as well as edge conditions.

- *Plot Frequency* - The contour plot tape is written every "plot frequency" steps.

- *Axial Plots* - Specify the surface number, point location, dependent and independent variables, as well as x/y scaling factors. Each of the axial plots will be written to a file 'graphxxx', where 'xxx' is the number of the axial plot.

i - surface number
j - point number

Y variable - Select one of the following for a dependent variable:

1. pressure
2. density
3. edge velocity
4. temperature
5. C_f (base on edge conditions)
6. C_f (base on reference conditions)
7. C_h (base on edge conditions)
8. C_h (base on reference conditions)
9. Momentum thickness
10. Displacement thickness
11. Energy thickness
12. Heating rate (B/(ft^2 sec), J/(m^2 sec))
13. Viscous loads; Here "i" is the surface number and 'j' is the load (1=x force, 2=y force, 3=z force, 4=x moment, 5=y moment, 6=z moment). To get total body viscous loads set "i" to 0.
14. Induced loads. Same options apply as for viscous loads.

Y scale - Dependent variable scale factor.

X variable - Select one of the following for an independent variable:

1. Axial location, z
2. Reynolds number (based on axial location), Re_z
3. Streamline length, s
4. Reynolds number (based on streamline length), Re_s

X scale - Independent variable scale factor.

- *Contour Plots* - Specify a list of target stations (separated by blank spaces) where the contour plots are desired (the files are written to 'graphcxxx', where 'xxx' refers to the appropriate contour file number). Then list the following parameters for each contour plot:

Surface - Enter the surface number. The surface numbers are counted sequentially along each of the four edges for each zone (i.e., Zone 2, Edge3 corresponds to surface number 7).

Variable - Select one of the following for a dependent variable:

1. pressure
2. density
3. edge velocity
4. temperature

5. C_f (base on edge conditions)
6. C_f (base on reference conditions)
7. C_h (base on edge conditions)
8. C_h (base on reference conditions)
9. Momentum thickness
10. Displacement thickness
11. Energy thickness
12. Heating rate (B/(ft^2 sec), J/(m^2 sec))

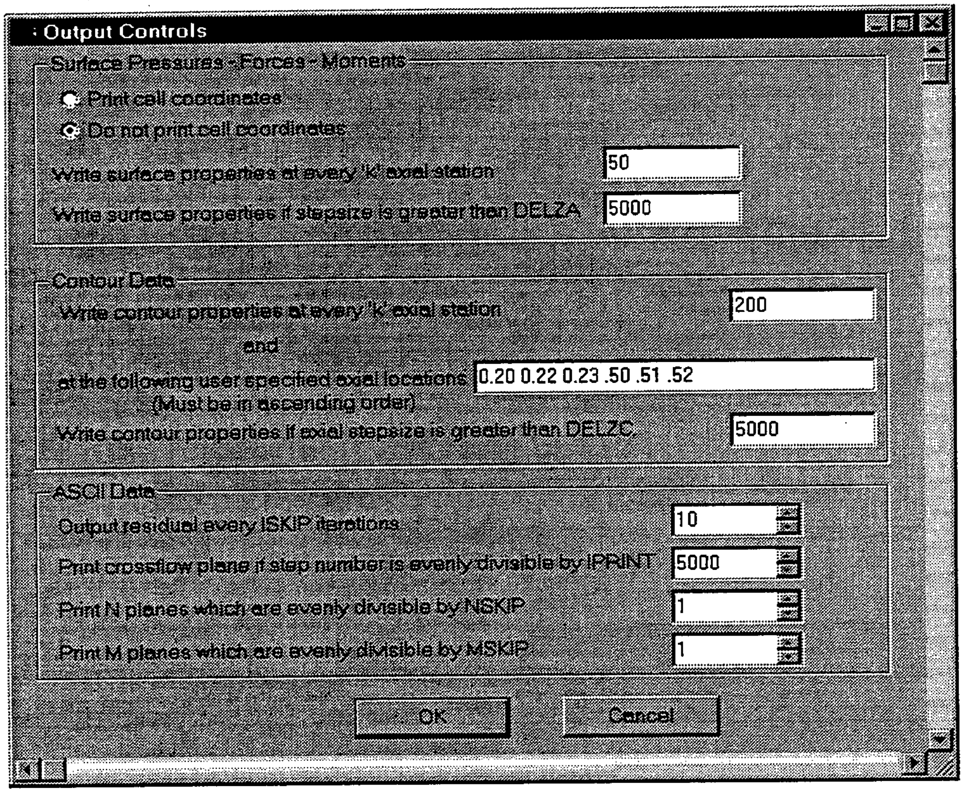
y-scale - Dependent variable scale factor.

x-scale - Crossflow variable scale factor.

4.5 OUTPUT CONTROLS

4.5.1 Parameter Description

This section controls which type of output the Zeus Euler solver generates, as well as the output frequency. The first section in the figure below controls the output for the surface pressures, forces, moments, and centers of pressure. The second section controls the output for generating contour plots of the flowfield variables. The final section controls both the ASCII output and screen residual.



Output Controls

Surface Pressures - Forces - Moments

○ Print cell coordinates

● Do not print cell coordinates

Write surface properties at every 'k' axial station: 50

Write surface properties if stepsize is greater than DELZA: 5000

Contour Data

Write contour properties at every 'k' axial station: 200

and

at the following user specified axial locations (Must be in ascending order): 0.20 0.22 0.23 .50 .51 .52

Write contour properties if axial stepsize is greater than DELZC: 5000

ASCII Data

Output residual every ISKIP iterations: 10

Print crossflow plane if step number is evenly divisible by IPRINT: 5000

Print N planes which are evenly divisible by NSKIP: 1

Print M planes which are evenly divisible by MSKIP: 1

FIGURE 30. OUTPUT CONTROL PARAMETERS

- *Surface pressures, forces, and moments* – This section handles the output for the surface pressure, forces, and moments.

- *Print cell coordinates* – Write the cell coordinates (axial and radial location) at each cell where the data is written along with the surface pressure, forces, moments, and centers of pressure.

- *Write surface properties* – Specify both an incremental value and a maximum stepsize for generating the output data. The Zeus^{++} code writes the output at every specified incremental value (axially) as well as if the axial stepsize is larger than the specified parameter.

- *Contour data* – Contour plots of a variety of flowfield parameters (pressure, density, Mach number, etc…) can be generated at various axial stations.

- *'k' Axial stations* – Enter the increment for generating the contour plot output data. The code will write the output data at every user specified incremental axial station.

- *User defined stations* – Specify the specific axial locations where the contour data is desired. This is useful for generating contour plots at specific locations in the flowfield (i.e., leading/trailing edges of control surfaces).

- *Given stepsize* – The Zeus^{++} code generates a contour plot file every time the axial stepsize is greater than the specified value.

- *ASCII data* – The ASCII data is written to the 'fort.9' data file in the 'filename' directory.

- *Output residual* – Specify the axial step increment for writing out the residual data. Note, the code writes the residual data to the screen as well as to the 'fort.9' data file.

- *Crossflow plane* – Specify the axial increment for saving the ASCII data from a crossflow plane (i.e., an axial station).

- *Print 'M', 'N' planes* – Specify the radial and circumferential increments for saving the ASCII crossflow plane data (i.e., Write out the data at every 'm' radial points and 'n' circumferential points). The computational effort required for I/O can be minimized by not writing out every radial and circumferential data point.

CHAPTER 5: CODE EXECUTION

OVERVIEW

This section is used to select which cases are evaluated as well as what type of solution is performed. A step by step procedure for executing a run is provided below along with a sample view of the execution screen.

- Select the desired run.

Note: Multiple runs may be selected in a number of ways. The first is to hold down the 'Ctrl' key and click on each of the desired runs. Alternatively, you can click on the first desired run, hold down the 'shift' key, and then click on the last run to select all cases in-between. Finally, you can click on the 'select All' button to run all available cases.

- Select the type of run desired.

Note: If 'Full Run' is selected the code will execute the Inviscid Euler solver, write out the flowfield at the axial intervals specified in the Boundary Layer Execution Parameter section, and then execute the boundary layer solver to get an approximate viscous solution. If 'Inviscid Run' is selected the code will execute the Inviscid Euler solver and then terminate. No viscous approximation will be calculated. If a viscous calculation is desired, click on the 'Viscous Run' button, and select the desired case. Only the runs that have been solved with the inviscid solver will show up as choices when the 'Viscous Run' button is selected. This is because the viscous boundary layer solver requires the inviscid flowfield as a boundary condition.

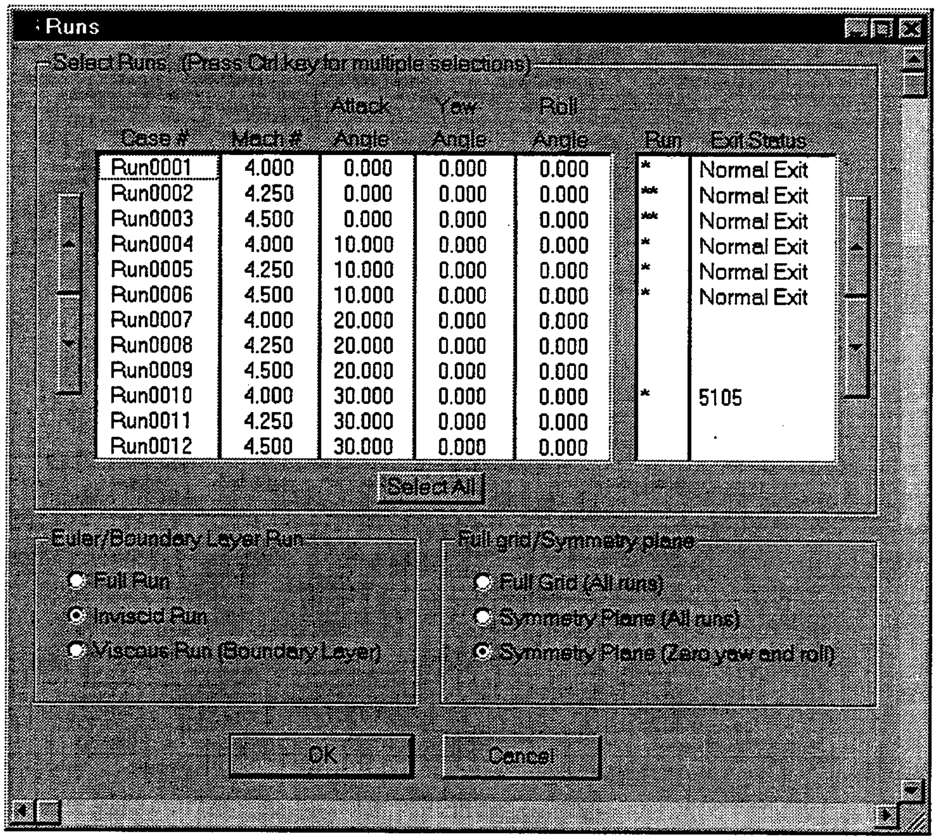
- Specify whether a symmetry plane exists.

Note: If you know a priori that the geometry is non-symmetric about the pitch plane, you must select 'Full Grid (All Runs)'. If the geometry is symmetric about the pitch plane choose either 'symmetry Plane (All Runs)' if you will not be running any asymmetric cases, or alternatively, choose 'symmetry Plane (Zero Roll and Yaw)' to run a symmetry plane only for those cases with zero yaw and roll angles. If a symmetry plane is being considered then only half of the computational domain (180 vs. 360 degrees) is considered.

Running with a symmetry plane will cut the computational time approximately in half.

- Click on 'OK' to perform the desired calculations.

- After the runs have completed, revisit the 'Execution' section to check on the termination status of each run. If the 'Runs' column contains one asterisk (*) then the inviscid solution has completed (e.g., Run0001). If it contains double asterisks (**) then both the inviscid and viscous solutions have been completed (Run0002). In addition, the 'Exit Status' column should contain 'Normal Exit' if no runtime errors were encountered. If the code did not exit normally then an error code will be provided. These error codes are 5000 series numbers for the Euler solver, 6000 series numbers for the boundary layer solver, 7000 series numbers for the cone solver, and 8000 series numbers for the blunt body solver. A description of each of the possible error codes is presented in the Error Code section.



Case #	Mach #	Attack Angle	Yaw Angle	Roll Angle	Run	Exit Status
Run0001	4.000	0.000	0.000	0.000	*	Normal Exit
Run0002	4.250	0.000	0.000	0.000	**	Normal Exit
Run0003	4.500	0.000	0.000	0.000	**	Normal Exit
Run0004	4.000	10.000	0.000	0.000	*	Normal Exit
Run0005	4.250	10.000	0.000	0.000	*	Normal Exit
Run0006	4.500	10.000	0.000	0.000	*	Normal Exit
Run0007	4.000	20.000	0.000	0.000		
Run0008	4.250	20.000	0.000	0.000		
Run0009	4.500	20.000	0.000	0.000		
Run0010	4.000	30.000	0.000	0.000	*	5105
Run0011	4.250	30.000	0.000	0.000		
Run0012	4.500	30.000	0.000	0.000		

Euler/Boundary Layer Run
○ Full Run
● Inviscid Run
○ Viscous Run (Boundary Layer)

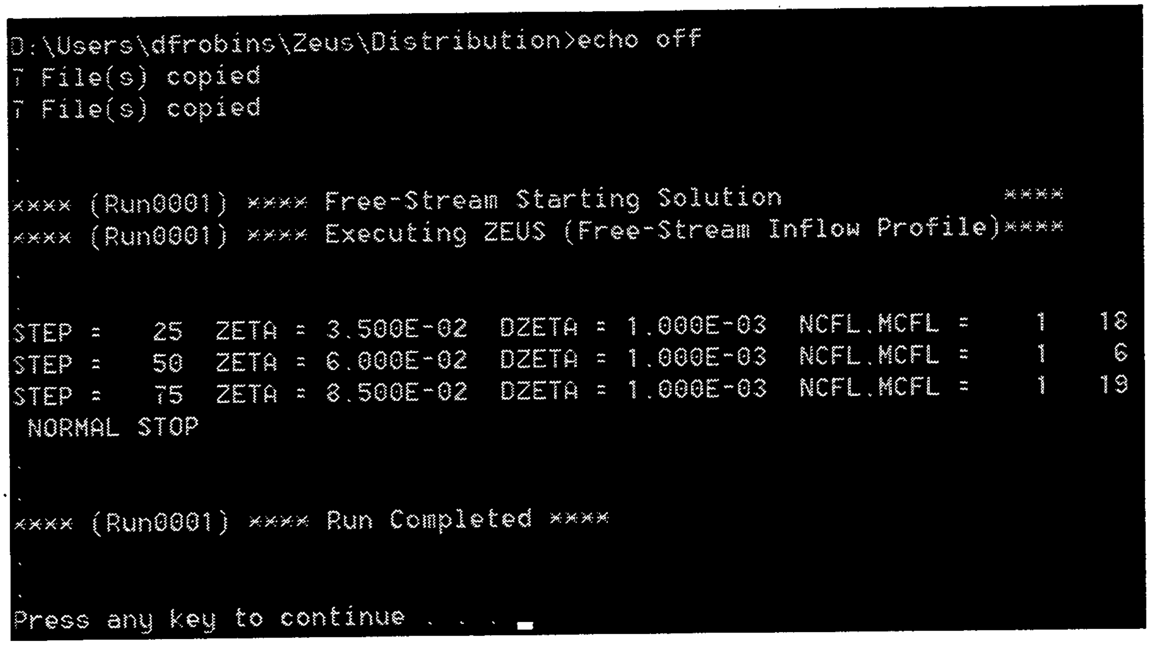
Full grid/Symmetry plane
○ Full Grid (All runs)
○ Symmetry Plane (All runs)
● Symmetry Plane (Zero yaw and roll)

FIGURE 31. EXECUTIVE CONTROL PARAMETERS

5.1 CONSOLE OUTPUT

5.1.1 Euler Console Output

As the Euler solver is executing, a variety of information is written to a command prompt (see sample below). This includes messages denoting which code is currently running (cone, blunt body, or Euler solver) as well as a number of parameters denoting the status of the run.



```
D:\Users\dFrobins\Zeus\Distribution>echo off
7 File(s) copied
7 File(s) copied
.
.
xxxx (Run0001) xxxx Free-Stream Starting Solution               ****
xxxx (Run0001) xxxx Executing ZEUS (Free-Stream Inflow Profile)****
.
.
STEP =    25  ZETA = 3.500E-02  DZETA = 1.000E-03  NCFL.MCFL =     1   18
STEP =    50  ZETA = 6.000E-02  DZETA = 1.000E-03  NCFL.MCFL =     1    6
STEP =    75  ZETA = 8.500E-02  DZETA = 1.000E-03  NCFL.MCFL =     1   19
 NORMAL STOP
.
.
xxxx (Run0001) xxxx Run Completed ****
.
.
Press any key to continue . . .
```

FIGURE 32. EULER CONSOLE OUTPUT

The output written to the command prompt for the inviscid solver includes the following parameters:

- *Starting profile* - Denotes which type of starting solution is employed (freestream, cone, or blunt body). This parameter in specified in the Integration Control section.

- *Step* - Counter for the number of axial steps taken. The frequency with which the screen data is written is controlled by the Output Residual parameter in the Output Control section.

- *Zeta* - Axial location of the current integration step.

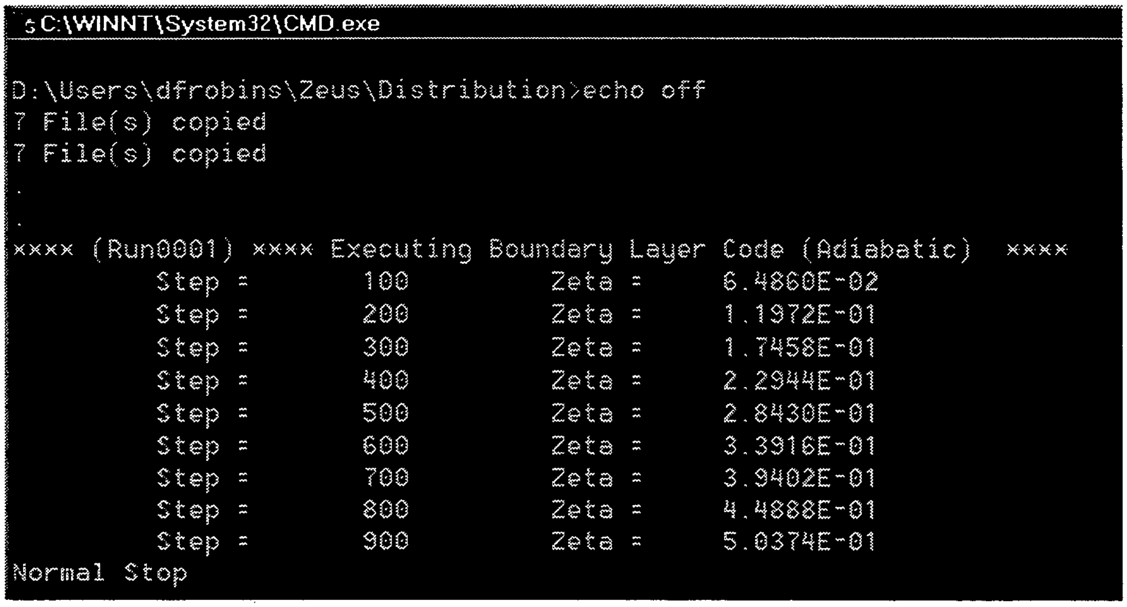
- *dZeta* - Local axial stepsize. This value changes as the solution proceeds downstream and is controlled by the stability criterion specified in the Integration Control section. If the axial stepsize is too large to resolve the geometric features then the maximum allowable stepsize can be adjusted in the Integration control section.

- *nCFL, mCFL* - The radial and circumferential index for the maximum residual.

- *Exit Status* - Upon termination the exit status of the current run is displayed along with any error messages that occurred during the Euler solution. 'Normal Stop' is displayed if no runtime errors were encountered.

  Note: After the run has completed the window will display 'Press Any Key' to continue. When a key is depressed, the command window is deleted and control is returned to the Zeus^{++} graphical user interface window. Whether or not the window pauses after completion is controlled by the 'Pause After Each Run' parameter in the Options section.

5.1.2 Boundary Layer Console Output



```
D:\Users\dfrobins\Zeus\Distribution>echo off
7 File(s) copied
7 File(s) copied
.
.
xxxx (Run0001) xxxx Executing Boundary Layer Code (Adiabatic)  xxxx
          Step =        100         Zeta =     6.4860E-02
          Step =        200         Zeta =     1.1972E-01
          Step =        300         Zeta =     1.7458E-01
          Step =        400         Zeta =     2.2944E-01
          Step =        500         Zeta =     2.8430E-01
          Step =        600         Zeta =     3.3916E-01
          Step =        700         Zeta =     3.9402E-01
          Step =        800         Zeta =     4.4888E-01
          Step =        900         Zeta =     5.0374E-01
Normal Stop
```

FIGURE 33. BOUNDARY LAYER CONSOLE OUTPUT

The output written to the command prompt for the viscous boundary layer solver includes the following parameters:

- *Case description* - During each run the case number being executed is displayed (Runxxxx) along with the appropriate heat flux boundary condition at the surface (i.e., Adiabatic or Isothermal). The boundary condition is specified in the General Aerodynamic Data section.

- *Step* - Counter for the number of axial steps taken. The frequency with which the screen data is written is controlled by the Output Residual parameter in the Output Control section.

- *Zeta* - Axial location of the current integration step.

- *Exit Status* - Upon termination the exit status of the current run is displayed along with any error messages that occurred during the Euler solution. 'Normal Stop' is displayed if no runtime errors were encountered.

  Note: After the run has completed the window will display 'Press Any Key' to continue. When a key is depressed, the command window is deleted and control is returned to the Zeus^{++} graphical user interface window. Whether or not the window pauses after completion is controlled by the 'Pause After Each Run' parameter in the Options section.

5.2 ERROR MESSAGES

Overview

The possible error messages encountered while executing the Zeus^{++} code are listed below for each section of the tool. Note that 5000 series errors occur in the Euler solver, 6000 series in the boundary layer solver, 7000 series in the cone solver, and 8000 series in the blunt body solver. A description for each series is listed in the corresponding section below.

Zeus Code	Error messages for the Zeus Euler code (inviscid solution).
Boundary Layer Code	Error messages for the Boundary Layer code (viscous solution).
Cone Code	Error messages for the Cone code (generates starting profiles).
Blunt Body Code	Error messages for the Blunt Body code (generates starting profiles).

5.2.1 Zeus Error Messages

5101 - 5104	Maximum iteration count to locate corner #1-#4 exceeded.
5105 - 5106	Negative pressure encountered while decoding conservative variables.
5108	Undefined surface geometry parameter.
5109	Forward swept leading edges (control surfaces) not yet implemented.
5110	Rearward swept trailing edges (control surfaces) not yet implemented.
5111 - 5112	After first order fix the flow remains subsonic.
5115 - 5116	Negative square root occurred in the Prandtl-Mayer expansion calculation.
5118	Error in linear interpolation for shockwave.
5120	Dimensions are too large.
5121	MAZ() array not consistent with MA() array.
5123	Specified radial clustering not attainable for aft GASP grid.
5124	Undefined control surface input.
5128 - 5129	Clustering parameters not consistent between the Cone code and the Zeus code.
5130	Error in oblique shock calculation.
5131	No possible oblique shock solution.
5132	No convergence in Prandtl-Meyer calculation.
5134	The number of specified axial plotting stations is greater than the maximum allowed. The MaxPlots parameter must be increased in the parameter statement and the code must be recompiled.
5135	Invalid starting solution passed to the Euler solver. Typically caused by errors in the cone or blunt body starting profile generators.

5.2.2 Boundary Layer Error Messages

6001	Maximum iteration count reached
6002	Invalid axial force calculation

5.2.3 Cone Error Messages

7001	Shock angle too great. If excessive turning angles are encountered during the cone solution then the initial profile will be subsonic.

5.2.4 Blunt Body Error Messages

8001	Errors in the blunt body starting profile generator.

CHAPTER 6: POST-PROCESSOR

OVERVIEW

The Zeus^{++} tool includes a built-in post-processor for viewing the solution data. A variety of methods of examining the results are available including ASCII representations of the data as well as plots of a wide variety of parameters, all of which are discussed in detail below. The plotting features of the Zeus^{++} code make use of the Tecplot[9] plotting package.

Note: Zeus^{++} can be used if Tecplot is not installed on the users' machine, however, the plotting capabilities of the post-processor will not be functional. ASCII representations of the data will still be available.

The 'PostProcessor' menu option offers the following commands:

ASCII Forces/Moments	Examine an ASCII data file of the forces/moments and centers of pressure acting on the body.
Single Run Plots	Line plots of the surface pressures, forces, and moments. Vector and streamline plots. Contour plots of a variety of flowfield parameters (Mach number, density, temperature, ...etc.).
Multi-Run Plots	Comparative line plots of forces/moments versus a variety of independent variables (Mach number, angle of attack, yaw angle, and roll orientation).

6.1 ASCII Data

6.1.1 Forces & Moments

There are two options when examining the forces, moments, and centers of pressure for a given set of runs. The first is to view the totals or the sum from each of the zone edges. This gives the sum of the values from the missile surface and the control surfaces, or the total acting on the geometry. The second option prints the forces/moments separately on each edge of each zone. This is often useful if the values on a particular control surface are desired (i.e., force due to 1 fin). Recall, that edge$_1$ refers to the missile surface, edge$_2$ and edge$_3$ refer to the zonal boundaries (i.e., the control surfaces, if they exist), and edge$_4$ refers to the outer boundary (see figure below).

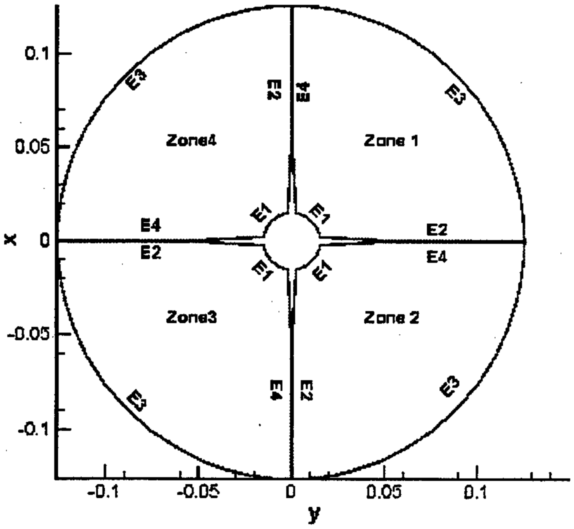


FIGURE 34. EULER GRID REFERENCE DIAGRAM

For each case run, the ASCII output file, 'ForcesMoments.txt' will contain the following data:

Mach #

Angle of Attack

Angle of Yaw

Roll Angle

Normal, Side, and Axial Force

Yaw, Pitch, and Roll Moments

X_{cp}/L_{Ref} and Y_{cp}/L_{Ref} (centers of pressure)

Note: An approximate value for the axial force due to the base (i.e., base drag) is calculated using the methodology of reference 14. Note that the corrections due to angle of attack and fin thickness have been omitted. Also, note that the base drag calculation requires the ratio of the base area to the reference area (A_{base}/A_{ref}). Zeus^{++} calculates the base area assuming a circular cross-section, and therefore, if a non-circular cross-sectional geometry is applied, the resultant base drag must be multiplied by the factor (A_{base}/A_{circ}). In other words, an equivalent circular base area must be applied.

If both the inviscid and viscous solutions have been performed then the output for the forces/moments will be listed componentwise. In other words, the inviscid, viscous, and induced contributions will be listed along with the totals.

6.2 SINGLE RUN PLOTS

Overview

This section of the post-processor is used to examine the output data for a single run. To make a comparative analysis of different runs see the Multi-Run Plots section.

The 'single Run Plots' menu offers the following three choices:

Surface Pressures	Line plot of the surface pressures.
Forces and Moments	Line plots of the forces, moments, and centers of pressure.
Contour/Vector Plots	Vector and streamline plots. Contour plots of a variety of flowfield parameters (Mach number, density, temperature, ...etc.).

6.2.1 Surface Pressures

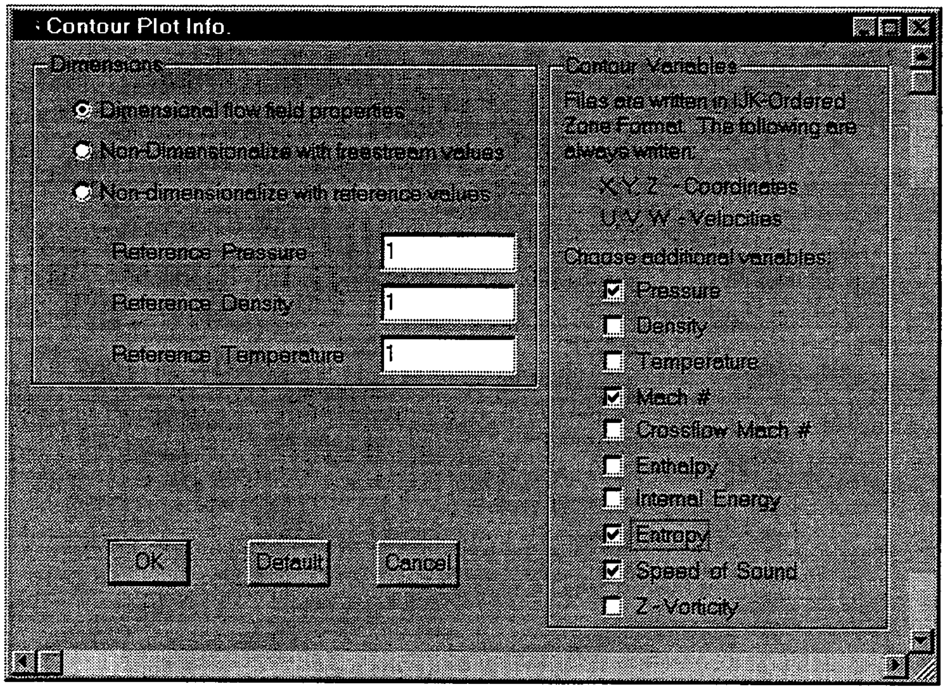
The first option, 'Plot Surface Pressure', will display a graph of the surface pressure versus axial distance for zone edges 1, 2, and 4 (see figure in ASCII data section). The pressures are listed for each circumferential plane on the missile surface and for each radial plane on the zone edges (i.e., control surfaces). Each plane is labeled as 'Plane #x, Zone #y, Edge #z'. If the Edge number is '1' then the Plane # refers to the circumferential index. If the Edge number is '2' or '4' then the Plane # refers to the radial index.

6.2.2 Forces and Moments

The second single-run plotting option, 'Plot Forces & Moments', will plot either the total force/moment or the edge force/moment as a function of axial location. The totals versus edge values are described in the ASCII data section. The 'x', 'y', 'z' coordinates refer to normal, side, and axial components, respectively.

6.2.3 Contour/Vector Plots

This plotting option generates contour plots of a variety of flowfield variables for a given run. Click on 'Choose Plot Attributes' to bring up the dialog box shown below and specify the plotting attributes. Select the dimensions and the variables desired for the contour plot. Note that the coordinates (x, y, z) and the velocities (u, v, w) are always written to the contour plot file. After the plot information has been specified, click 'OK' to accept the values. Then select 'Plot Data' to generate the contour plots.



Dimensions

● Dimensional flow field properties
○ Non-Dimensionalize with freestream values
○ Non-dimensionalize with reference values

Reference Pressure: 1
Reference Density: 1
Reference Temperature: 1

Contour Variables

Files are written in IJK-Ordered Zone Format. The following are always written:

X,Y,Z - Coordinates
U,V,W - Velocities

Choose additional variables:

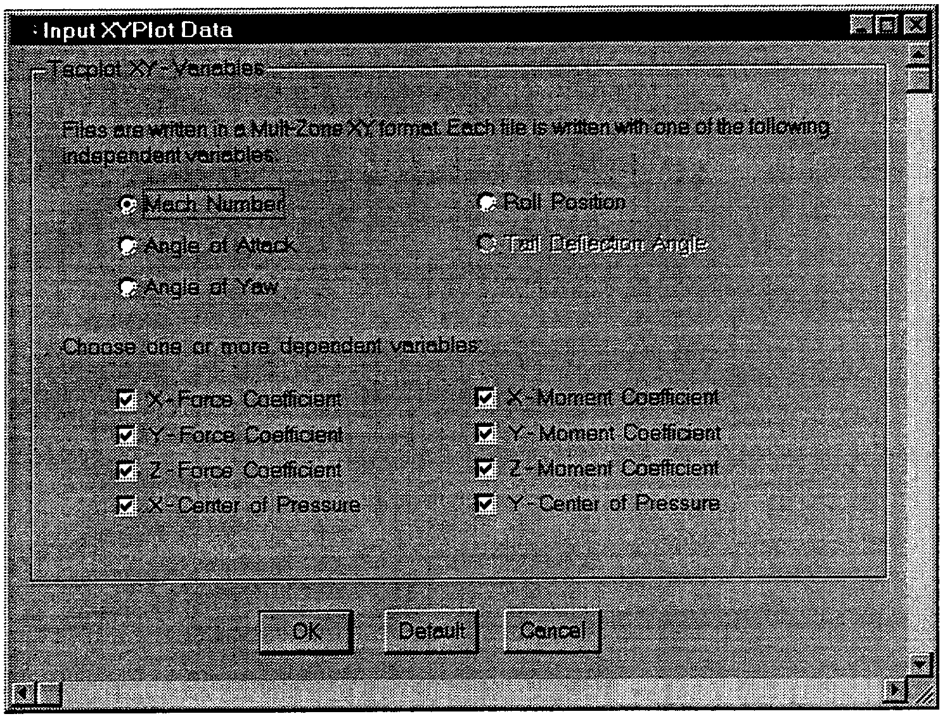
- ☑ Pressure
- ☐ Density
- ☐ Temperature
- ☑ Mach #
- ☐ Crossflow Mach #
- ☐ Enthalpy
- ☐ Internal Energy
- ☑ Entropy
- ☑ Speed of Sound
- ☐ Z - Vorticity

FIGURE 35. CONTOUR PLOT CONTROLS

6.3 MULTI-RUN PLOTS

Overview

This section of the post-processor is used to compare the effect of Mach number, angle of attack, yaw angle, and roll orientation on the forces, moments, and centers of pressure of a given geometry. Select the desired independent variable as well as the desired dependent variables (see figure below). Click on 'OK' to generate a plot for all of the completed runs.



Files are written in a Mult-Zone XY format. Each file is written with one of the following independent variables:

● Mach Number
○ Angle of Attack
○ Angle of Yaw
○ Roll Position
○ Tail Deflection Angle

Choose one or more dependent variables:

☑ X - Force Coefficient
☑ Y - Force Coefficient
☑ Z - Force Coefficient
☑ X - Center of Pressure
☑ X - Moment Coefficient
☑ Y - Moment Coefficient
☑ Z - Moment Coefficient
☑ Y - Center of Pressure

FIGURE 36. 24 PLOT CONTROLS

7.0 REFERENCES

1. GASP v3 Computational Fluid Dynamics Solver, Aerosoft, Inc., 1998.
2. Wardlaw, A.B.; Davis, S.F.; and Priolo, F.J., *A Second Order Gudonuv's Method for Supersonic Tactical Missile Computations*, NSWC TR 86-506, Dec 1998.
3. Hsieh, T. and Priolo, F.J., *Generation of the Starting Plane Flowfield for Supersonic Flow Over a Spherically Capped Body*, NSWC TR 84-484, May 1985.
4. Wardlaw, A.B. and Priolo, F.J., *Applying the Zeus Code*, NSWC TR-86-508, Dec 1986.
5. Mendenhall, Michael R., "Tactical Missile Aerodynamics: Prediction Methodology," Vol. 142, Progress in Astronautics and Aeronautics, 1992.
6. Wardlaw, A.B. and Baltakis, F.P., "An Integral Boundary Layer Procedure for Tactical Missiles," AIAA 92-1026, Aerospace Design Conference, Irvine, CA, Feb. 1992.
7. Wardlaw, A.B.; Priolo, F.J.; and Solomon, J.M., *A Multiple Zone Method For Supersonic Tactical Missiles*, NSWC TR 85-484, Jun 1986.
8. Wardlaw, A.B. and Priolo, F.J., Personal Communication and unpublished documentation.
9. Tecplot v7.0 Plotting Package, Amtec Engineering, 1998.
10. Hymer, T.C.; Moore, F.G.; and Downs, C., *User's Guide for an Interactive Personal Computer Interface for the Aeroprediction Code*, NSWC-DD TR 94-107, Jun 1994.
11. Baltakis, F.P.; Wardlaw, A.B.; Jr., Allen, J.M., "Leeside Crossflow Modeling in Euler Space Marching Computations," NSTC TR 86-342, Nov 1986.
12. Kwong, C.M. and Myring, D.F., "Fusiform Body Separation Flow Field Calculations using Euler and Boundary Layer Methods", Royal Aeronautics Society Conference on the Prediction and Exploitation of Separated Flows, Paper No. 27, Apr 1989.
13. Monta, J. W., "Supersonic Aerodynamic Characteristics of a Sparrow III Type Missile Model With Wing Controls and Comparison With Existing Tail-Control Results," NASA Technical Paper #1078, 1977.
14. Moore, F.G. and Hymer, T., "Improved Empirical Model for Base Drag Prediction on Missile Configurations Based on New Wind Tunnel Data," NSWC-DD TR 92-509, Oct 1992.
15. Shapiro, A.H., "The Dynamics and Thermodynamics of Compressible Fluid Flow," Vol. I, pp. 211-212, 1953.

APPENDIX A

TUTORIAL

A.1 INTRODUCTION

STARTING ZEUS++

Execution of the code begins by double clicking on the program 'ZeusPP.exe', entering a name for the current case ('tutorial'), and then clicking on 'start New Case'. For the tutorial, we will be considering a SeaSparrow missile[5,13] with both dorsal and tailfins.

Note: The name entered cannot contain blank spaces.

Execution of the Zeus^{++} code can be divided into four basic sections:

1. Geometry Setup
2. Case Inputs
3. Code Execution
4. Post-Processing

A.2 GEOMETRY SETUP

OVERVIEW

The geometry setup section can be sub-divided into four areas:

1) Specify the grid parameters
2) Define the missile geometry
3) Define the control surfaces
4) Generate the computational domain (optional).

A.2.1 GRID PARAMETERS

The properties that define the computational grid are specified in this section. Each of these parameters is described in detail in the Grid Parameters section.

- From the main menu, click on 'Geometry' and then select 'Grid Parameters' and 'Euler Grid Properties. For an explanation of each of the parameters on this screen, press the 'F1' key. The default values are adequate for the current tutorial, therefore, simply click on 'OK' to accept the current values.

Note: From the 'Grid Parameters' option, it is seen that there are two sets of grid parameters available. In the first (Euler Grid Properties), the parameters specified are applied during both the Zeus Euler solution as well as the viscous boundary layer solution. In the second (Navier-Stokes Grid Properties), the parameters specified are used to generate a grid, which can be exported to a full Navier-Stokes computational fluid dynamics solver. The Navier-Stokes parameters are used only for grid generation purposes, and therefore, if the user does not intend to export the grid to an N-S solver, this section may be omitted.

A.2.2 MISSILE SECTIONS

After specifying the grid parameters, the next step is to define the missile geometry. For the tutorial, the missile geometry will be entered as three separate sections: 1) Tangent-Ogive nose, 2) cylindrical mid-body and 3) boat-tail flare.

The first section is the Tangent-Ogive Nose

- Click on 'Geometry' and then select 'Missile Sections'.
- Click on the down arrow next to 'select Geometry Type' and select 'Tangent Ogive'.
- Click on 'Add New Section'.
- Enter the final axial location (0.0686) of the nose as well as the final radius (.01524). Note that the missile starts at 0,0.
- Click on 'OK' and the code will calculate and display the initial missile section.

The next section is the cylindrical mid-body and is described by a conic section.

- Click on the down arrow next to 'select Geometry Type' and then select 'Conic Section'.
- Click on 'Add New Section'.
- The initial axial location and radius are copied from the end of the previous section.
- Enter the final axial location (0.537) of the conic as well as the final radius (.01524).
- Click on 'OK'.

The final section is the boat-tail flare and is described by a conic section.

- Click on the down arrow next to 'select Geometry Type' and then select 'Conic Section'.

- Click on 'Add New Section'.
- The initial axial location and radius are copied from the end of the previous section.
- Enter the final axial location (0.5486) of the conic as well as the final radius (.013715).
- Click on 'OK'.
- Click on 'Finished'
- Press 'Ctrl-R' to regenerate the geometry.

A.2.3 CONTROL SURFACES

After specifying the missile geometry, the next step is to define the control surfaces. For the tutorial, two sets of control surfaces are considered. The first is the dorsal fins and the second is the tail fins. The control surfaces must be entered in the order in which they appear on the missile, therefore, the dorsal fins must be entered first, followed by the tail fins.

The first set of control surfaces (dorsal fins) will be generated using the 'Modified Wedge' option.

- Click on 'Geometry' and then select 'Control Surfaces'.
- Click on the arrow next to 'select Control Surface' and select 'Modified Wedge'.
- Click on 'Add Control Surface'.
- Enter a label for the control surface ("Dorsal Fins").
- Enter the axial location of the control surface hinge line ("0.2743"). The hinge line is the point on the control surface about which deflections are made.
- Enter the offset angles (in degrees) for each of the fins. For the current case the fins are located in the '+' configuration, and therefore, the offset angles are 90, 180, 270, and 360°.

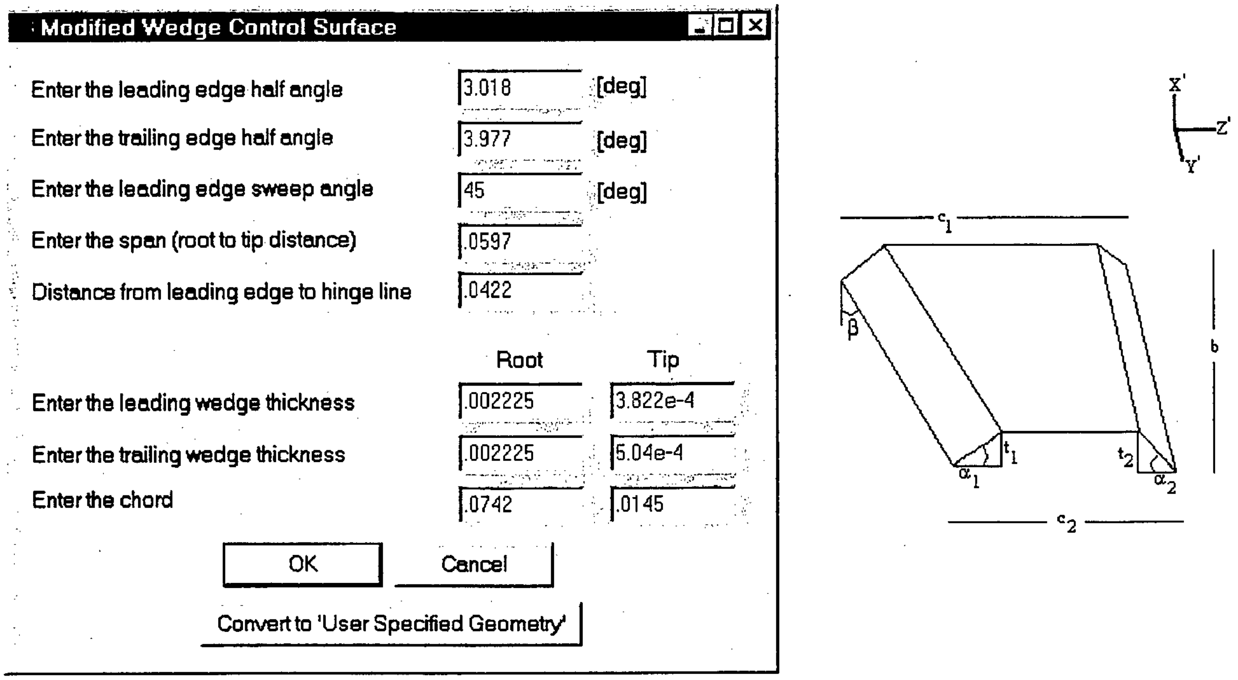
Note: It is possible to delete an individual fin by right clicking on the fin you wish to remove. Both the offset angle and the deflection angle will be replaced with '——' to indicate that they have been eliminated. In order to add the fin back into the calculation, simply right click on it a second time. This feature is useful when you want to have a different number of fins for each set of control surfaces (i.e., 2 dorsal fins and 4 tail fins).

- Enter the deflection angles (in degrees) for each of the fins. Use +5.0 for the 90 and 270° fins.

Note: The fins are always deflected in a clockwise orientation (i.e., entering +5 for both fins will generate an asymmetric deflection).

- Click 'OK'

- The parameters used to describe the modified wedge control surface are described in detail in the Modified Wedge section. For the Dorsal fins, enter the following values and click on 'OK' when completed:



Modified Wedge Control Surface

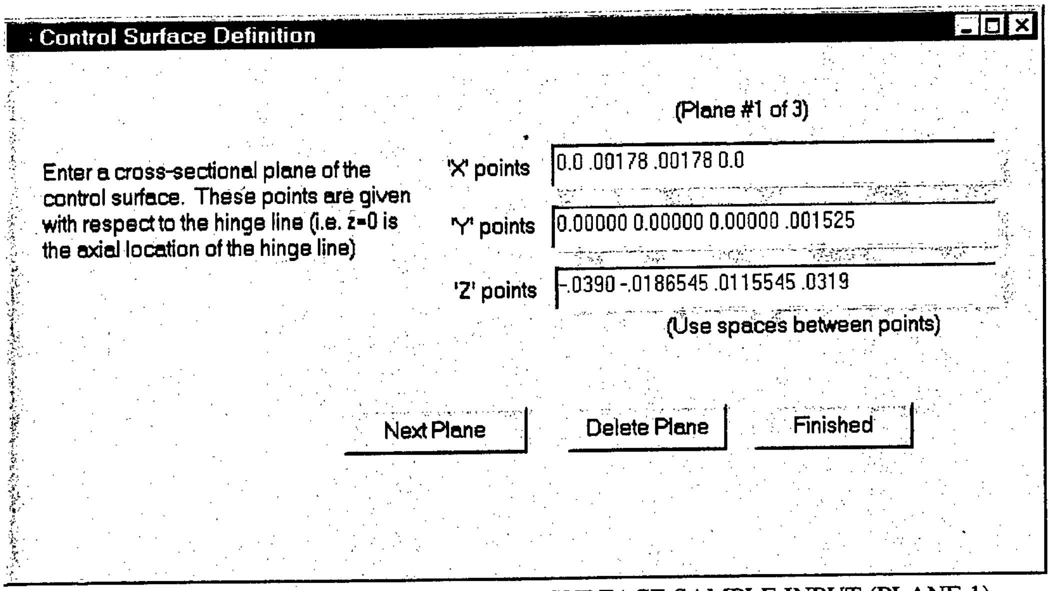
Enter the leading edge half angle: 3.018 [deg]
Enter the trailing edge half angle: 3.977 [deg]
Enter the leading edge sweep angle: 45 [deg]
Enter the span (root to tip distance): .0597
Distance from leading edge to hinge line: .0422

	Root	Tip
Enter the leading wedge thickness	.002225	3.822e-4
Enter the trailing wedge thickness	.002225	5.04e-4
Enter the chord	.0742	.0145

FIGURE A-1. MODIFIED WEDGE SAMPLE INPUT

The tailfins represent the second set of control surfaces. Because of the layout of the geometry of these fins, they will be generated using the 'User Defined Geometry' option.

- Click on the arrow next to 'select Control Surface' and select 'User Defined Geometry'.
- Click on 'Add New Section'.
- Enter a label for the control surface ("Tail Fins").
- Enter the axial location of the control surface hinge line ("0.5167").
- Enter the offset angles (in degrees) for each of the fins. For the current case the fins are located in the '+' configuration, and therefore, the offset angles are 90, 180, 270, and 360°.
- Enter the deflection angles (in degrees) for each fin (0°)
- Click 'OK'
- To ensure that the tail fins are described by planar surfaces the geometry must be entered as three cross sectional cuts. A detailed overview of the methodology used to generate this control surface geometry can be found at User Defined Geometry. For the tail fins under consideration, enter the following values for the three cross-sectional planes:



(Plane #1 of 3)

Enter a cross-sectional plane of the control surface. These points are given with respect to the hinge line (i.e. z=0 is the axial location of the hinge line)

'X' points 0.0 .00178 .00178 0.0

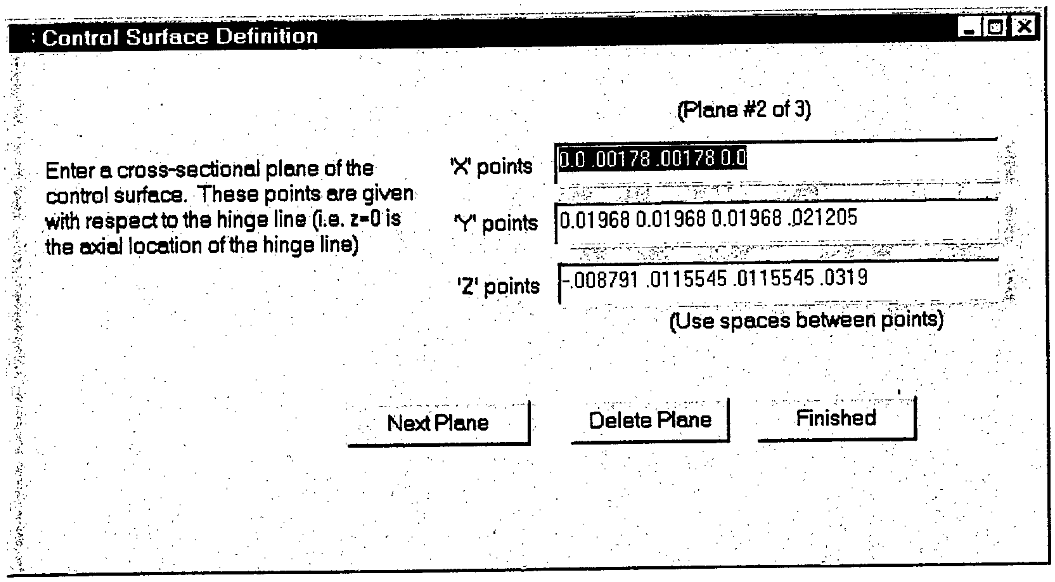
'Y' points 0.00000 0.00000 0.00000 .001525

'Z' points -.0390 -.0186545 .0115545 .0319

(Use spaces between points)

FIGURE A-2. USER DEFINED CONTROL SURFACE SAMPLE INPUT (PLANE 1)

- Click 'Next Plane'



(Plane #2 of 3)

Enter a cross-sectional plane of the control surface. These points are given with respect to the hinge line (i.e. z=0 is the axial location of the hinge line)

'X' points 0.0 .00178 .00178 0.0

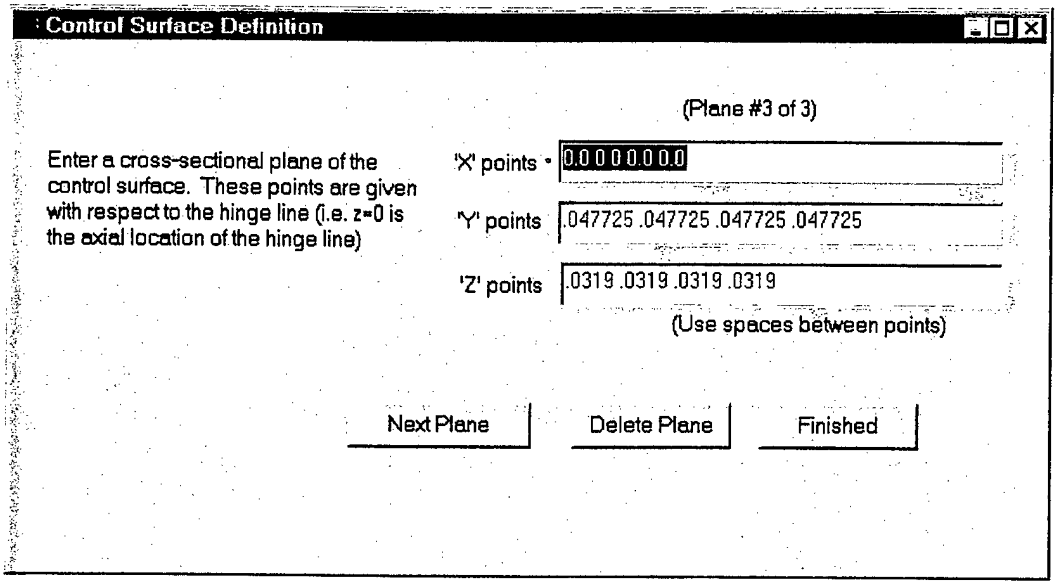
'Y' points 0.01968 0.01968 0.01968 .021205

'Z' points -.008791 .0115545 .0115545 .0319

(Use spaces between points)

FIGURE A-3. USER DEFINED CONTROL SURFACE SAMPLE INPUT (PLANE 2)

- Click 'Next Plane'



(Plane #3 of 3)

Enter a cross-sectional plane of the control surface. These points are given with respect to the hinge line (i.e. z=0 is the axial location of the hinge line)

'X' points 0.0 0.0 0.0 0.0

'Y' points .047725 .047725 .047725 .047725

'Z' points .0319 .0319 .0319 .0319

(Use spaces between points)

FIGURE A-4. USER DEFINED CONTROL SURFACE SAMPLE INPUT (PLANE 3)

- Click 'Finished'
- Click 'Finished'
- Press 'Ctrl-R' to regenerate the geometry

A.2.4 GRID GENERATION (OPTIONAL)

- View Euler Grid

The purpose of this section is to examine the inviscid grid that will be used by the Zeus Euler solver to ensure that an acceptable grid is being employed.

Note: 1) The axial step size for the inviscid solution is calculated at runtime, however, the axial step size for grid generation purposes is taken to be constant.
2) The outer boundary, for grid generation purposes, is taken to be that specified in the Euler grid parameter section.

- Click on 'Geometry' and then select 'Grid Generation' and 'View Euler Grid'
- Click 'Plot Grid as a Series of 2d planes'
- Enter a starting axial location (0.01)
- Enter a final axial location (0.5486)
- Enter the number of axial step (25)
- Click 'OK'
- After the Tecplot window appears:
- Click 'File', 'Macro', and then 'Play'
- Double click on the '2dGrid_Movie.mcr' file in the 'Playback' directory.

Note: The macro file will step axially down the missile and take a snapshot of each location. The viewport shown during the movie generation (i.e., x and y-axis ranges) can be specified in Zeus^{++} before generating the Euler grid by clicking on 'View' and then selecting 'Options'. Enter the desired values for the 'x' and 'y' axis ranges. A detailed description of all of the parameters is provided in the View/Options section.

- A movie of the grid will be created and written out as 'Playback\Zeus_grid2d.rm'. This movie file can be replayed using the Tecplot Framer program.
- Navier-Stokes Grid

The Zeus^{++} code functions both as an inviscid solver (with approximate viscous corrections), and a CFD grid generator. This allows one to export a three-dimensional computational grid and the appropriate boundary conditions to a full Navier-Stokes solver. Currently, the grid and boundary condition files generated are suitable for the GASP flow solver. One circumferential plane of the Seasparrow Navier-Stokes grid is shown below with an explanation of each parameter provided in the Navier-Stokes Grid section.

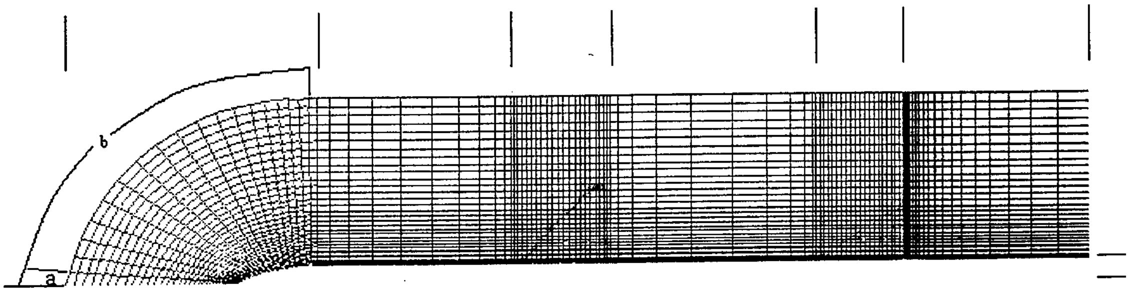


FIGURE A-5. SAMPLE OUTPUT OF NAVIER-STOKES GRID GENERATOR (2d CROSS-SECTION SHOWN)

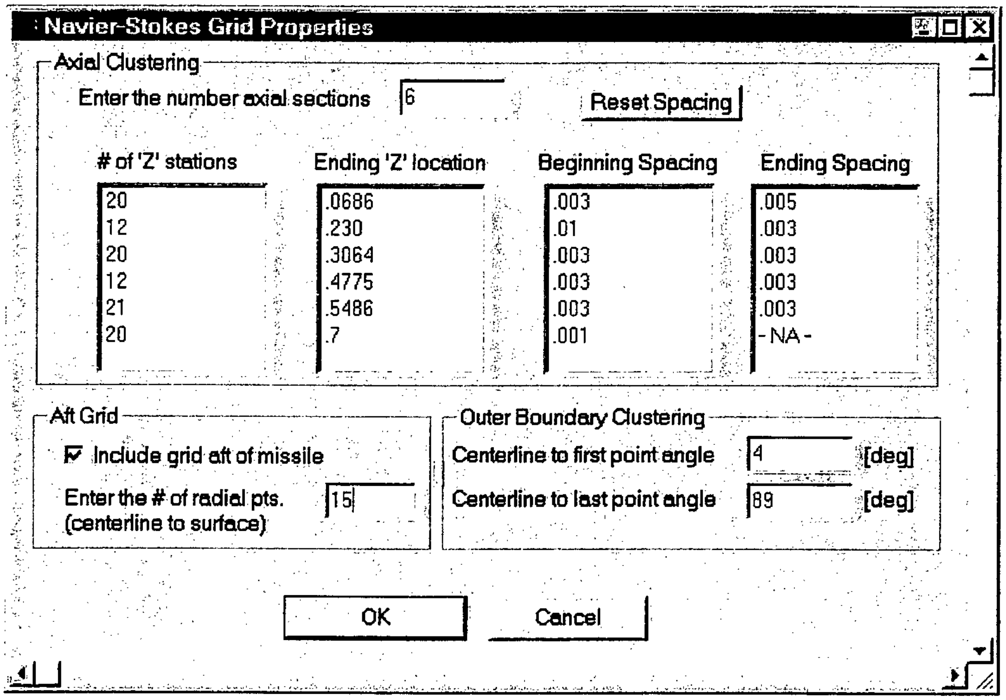
- Click on 'Geometry', 'Grid Generation', 'Navier-Stokes Grid', and 'Generate Grid'
- Enter the number of axial sections (6). The geometry is separated into a number of axial stations so that clustering can be performed at a number of axial locations. From the above figure, the six axial sections are shown between the vertical lines. The six sections are as follows:

1. Nose section
2. Section between nose and first set of fins
3. Section covering dorsal fins
4. Section between dorsal fins and tail fins
5. Section covering the tail fins
6. Section aft of the missile

Note: If you extend the grid beyond the aft end of the missile (shown in blue between the vertical lines), then an additional section will be included to calculate the flow directly behind the missile (shown in yellow between the horizontal lines).

- Enter the number of axial stations, ending axial location, beginning spacing, and ending spacing for each of the six sections (values provided below):

Note: To simplify the initial estimate for axial spacing a 'Reset Spacing' button is provided. Simply enter the number of axial stations and the ending axial location for each of the desired sections. Then, click on 'Reset Spacing' to obtain an equally spaced axial grid. This grid can then be modified to provide the desired axial clustering.



Axial Clustering

Enter the number axial sections 6

# of 'Z' stations	Ending 'Z' location	Beginning Spacing	Ending Spacing
20	.0686	.003	.005
12	.230	.01	.003
20	.3064	.003	.003
12	.4775	.003	.003
21	.5486	.003	.003
20	.7	.001	-NA-

Aft Grid

☑ Include grid aft of missile

Enter the # of radial pts. (centerline to surface) 15

Outer Boundary Clustering

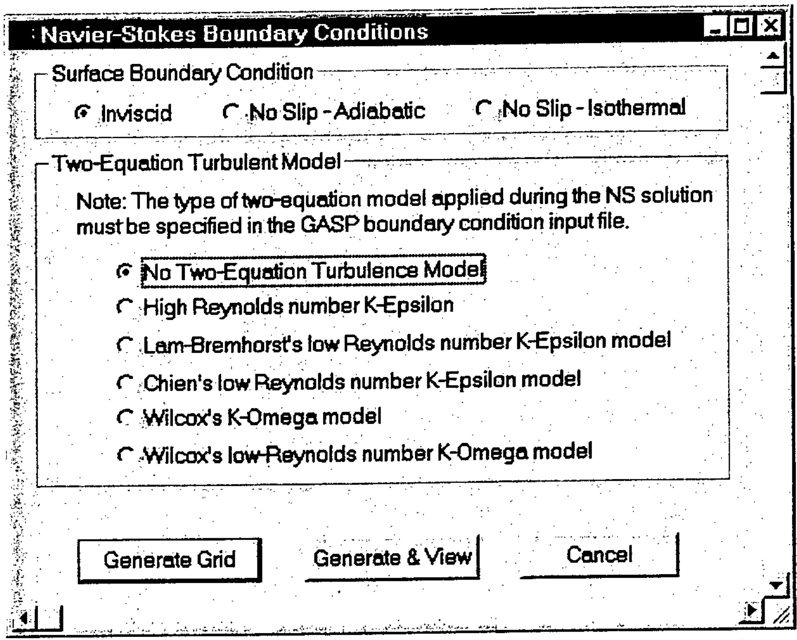
Centerline to first point angle 4 [deg]

Centerline to last point angle 89 [deg]

FIGURE A-6. SAMPLE INPUTS FOR NAVIER-STOKES GRID GENERATOR

- For the aft grid section, select the 'Include Grid Aft of Missile' and enter the number of radial points to include (15).
- In order to control the clustering of the outer boundary of the nose, the beginning and ending angles (see 'a' and 'b' in the above figure) of the outer boundary are specified. Enter the values shown in the above figure.
- Click on 'Ok' to bring up the boundary condition window (see Figure A-7).
- Select the surface boundary condition.
- Select the two-equation turbulence model.
- Click on 'OK' to generate the computational grid and boundary condition files.

Note: The output from the grid generation code is placed in the 'filename/RunGrid' directory. For the current case, this is 'Tutorial/RunGrid'. The two files are the grid geometry in Plot3d format (GASPGrid.p3da) and the GASP boundary condition input file (GASPbc.inp).



Surface Boundary Condition

● Inviscid ○ No Slip - Adiabatic ○ No Slip - Isothermal

Two-Equation Turbulent Model

Note: The type of two-equation model applied during the NS solution must be specified in the GASP boundary condition input file.

● No Two-Equation Turbulence Model
○ High Reynolds number K-Epsilon
○ Lam-Bremhorst's low Reynolds number K-Epsilon model
○ Chien's low Reynolds number K-Epsilon model
○ Wilcox's K-Omega model
○ Wilcox's low-Reynolds number K-Omega model

FIGURE A-7. SAMPLE BOUNDARY CONDITION INPUTS FOR NAVIER-STOKES GRID GENERATOR

A.3 INPUT PARAMETERS

OVERVIEW

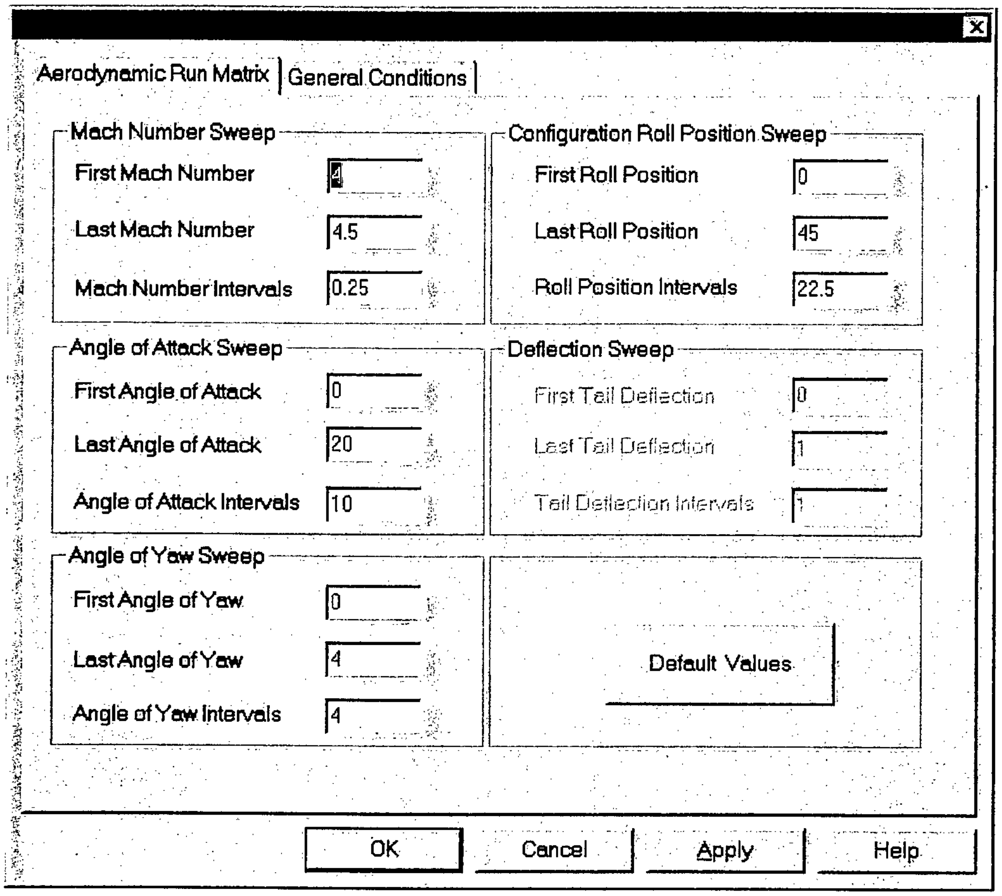
After the computational domain, missile, and control surface geometries have been specified, the next step is to input the following parameters:

1. Aerodynamic Data
2. Integration Controls
3. Separation Modeling Parameters
4. Boundary Layer Controls
5. Output Controls

A.3.1 AERODYNAMIC DATA

This section is used to specify the Mach numbers and missile orientations (angle of attack, yaw, and roll) to be used during execution of the flow solver. Rather than specifying singular values, sweeps are defined for each of the above mentioned parameters. In order to generate a given sweep, input the beginning, ending, and incremental value for the desired parameter (see below).

- Click on 'Inputs' and then select 'Aerodynamic data'.
- Select the 'Aerodynamic Run Matrix' tab and then enter the values shown in the figure below.



Mach Number Sweep
First Mach Number 4
Last Mach Number 4.5
Mach Number Intervals 0.25

Configuration Roll Position Sweep
First Roll Position 0
Last Roll Position 45
Roll Position Intervals 22.5

Angle of Attack Sweep
First Angle of Attack 0
Last Angle of Attack 20
Angle of Attack Intervals 10

Deflection Sweep
First Tail Deflection 0
Last Tail Deflection 1
Tail Deflection Intervals 1

Angle of Yaw Sweep
First Angle of Yaw 0
Last Angle of Yaw 4
Angle of Yaw Intervals 4

FIGURE A-8. SAMPLE OF AERODYNAMIC RUN MATRIX PARAMETERS

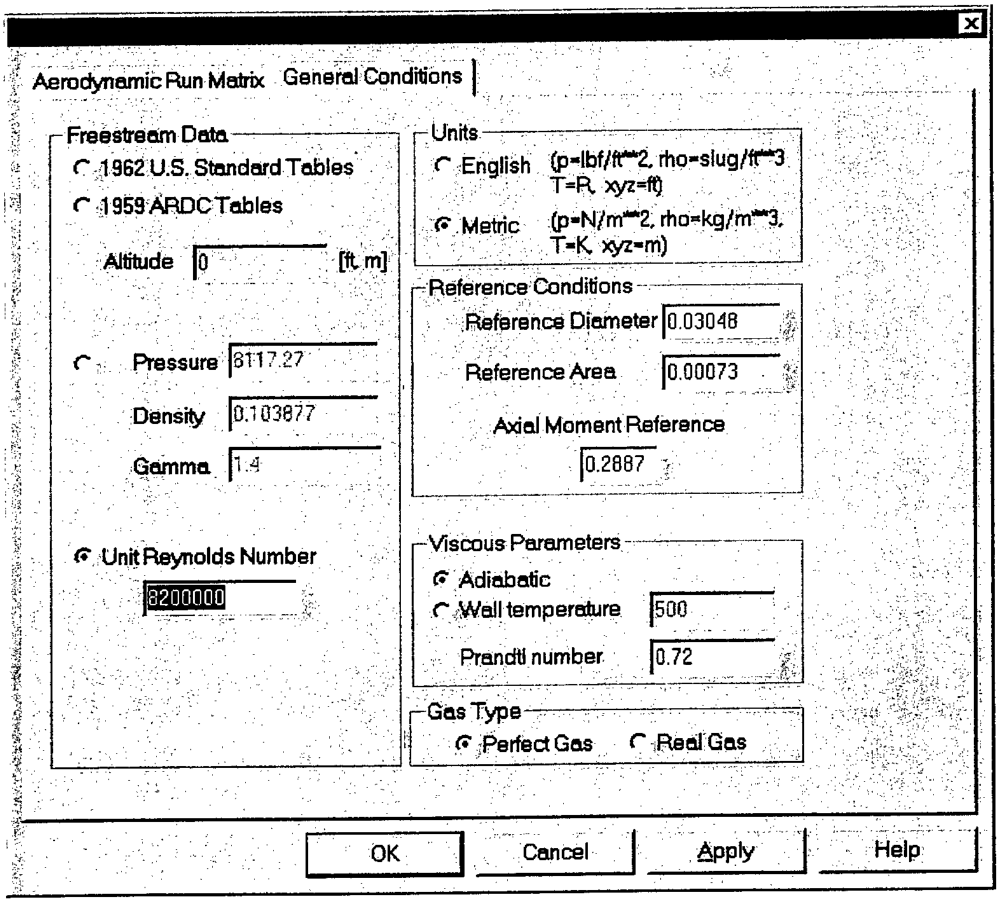
The next section is used to specify the remaining aerodynamic parameters including free-stream atmospheric conditions, units, reference conditions (diameter, area, and moment arm), viscous parameters (wall heat flux boundary condition, Prandtl number), as well as the gas type (real or ideal gas).

- Click on the 'General Conditions' tab to specify the remaining aerodynamic parameters.

- The free-stream aerodynamic parameters (density, pressure, and ratio of specific heats) can be specified in a variety of ways. For the current example, a unit Reynolds number is provided and the free-stream quantities are computed. Alternate methods include explicitly specifying the values or specifying an altitude and using standard atmospheric tables to look up the corresponding values.
- Specify the units as metric.
- Enter the reference diameter, area, and moment arm shown in the figure below.
- Select 'Adiabatic' and specify a Prandtl number of 0.72.

Note: The viscous parameters are not used in the Zeus Euler solver. These values are, however, required inputs for the approximate viscous (boundary layer) solver. Therefore, these values are not required unless you are running the viscous solution along with the inviscid flow solver.

- Click on 'OK' and wait for the code to generate the required run matrix.



Freestream Data
○ 1962 U.S. Standard Tables
○ 1959 ARDC Tables
Altitude 0 [ft, m]
○ Pressure 8117.27
Density 0.103877
Gamma 1.4
● Unit Reynolds Number 8200000

Units
○ English (p=lbf/ft**2, rho=slug/ft**3 T=R, xyz=ft)
● Metric (p=N/m**2, rho=kg/m**3, T=K, xyz=m)

Reference Conditions
Reference Diameter 0.03048
Reference Area 0.00073
Axial Moment Reference 0.2887

Viscous Parameters
● Adiabatic
○ Wall temperature 500
Prandtl number 0.72

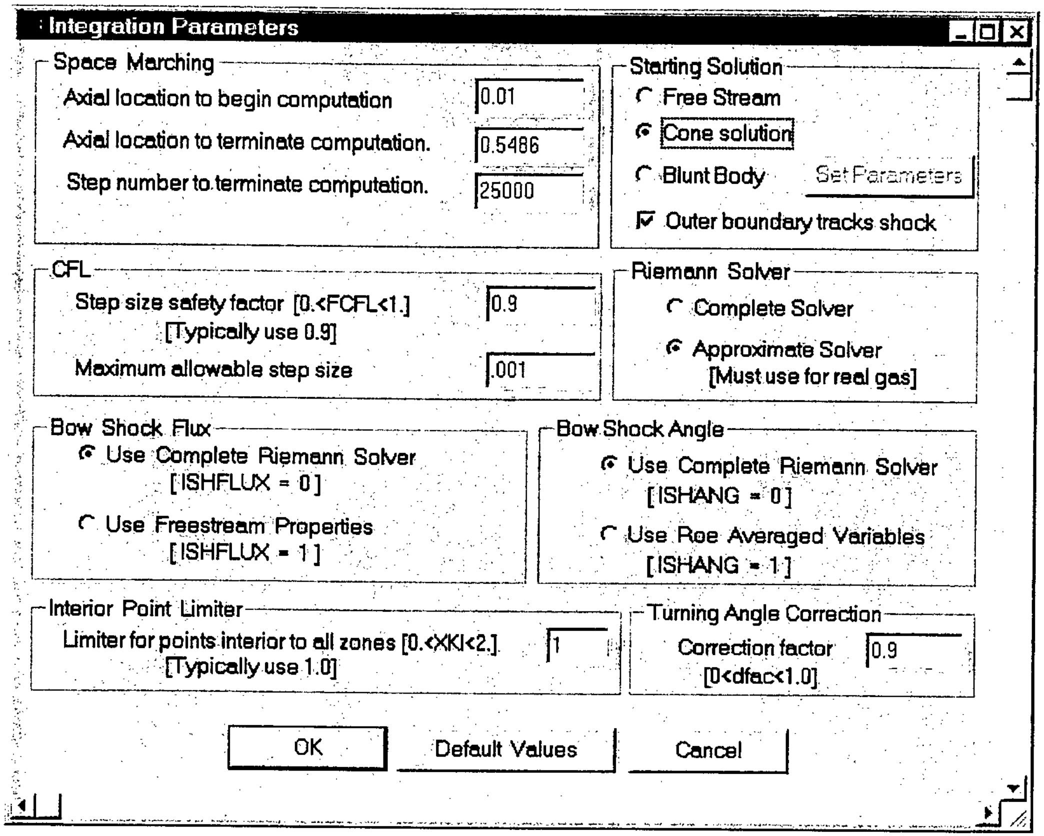
Gas Type
● Perfect Gas ○ Real Gas

FIGURE A-9. SAMPLE INPUTS FOR GENERAL AERODYNAMIC PARAMETERS

A.3.2 INTEGRATION CONTROL

This section is used to specify integration parameters, all of which are described in detail in the Integration Parameters section.

- Click on 'Inputs' and then select 'Integration Control'
- Enter all of the values shown in the figure below and click 'OK'.



Integration Parameters

Space Marching
Axial location to begin computation: 0.01
Axial location to terminate computation. 0.5486
Step number to terminate computation. 25000

Starting Solution
○ Free Stream
● Cone solution
○ Blunt Body
☑ Outer boundary tracks shock

CFL
Step size safety factor [0.<FCFL<1.] [Typically use 0.9]: 0.9
Maximum allowable step size: .001

Riemann Solver
○ Complete Solver
● Approximate Solver [Must use for real gas]

Bow Shock Flux
● Use Complete Riemann Solver [ISHFLUX = 0]
○ Use Freestream Properties [ISHFLUX = 1]

Bow Shock Angle
● Use Complete Riemann Solver [ISHANG = 0]
○ Use Roe Averaged Variables [ISHANG = 1]

Interior Point Limiter
Limiter for points interior to all zones [0.<XKI<2.] [Typically use 1.0]: 1

Turning Angle Correction
Correction factor [0<dfac<1.0]: 0.9

FIGURE A-10. SAMPLE INPUT FOR INTEGRATION PARAMETERS

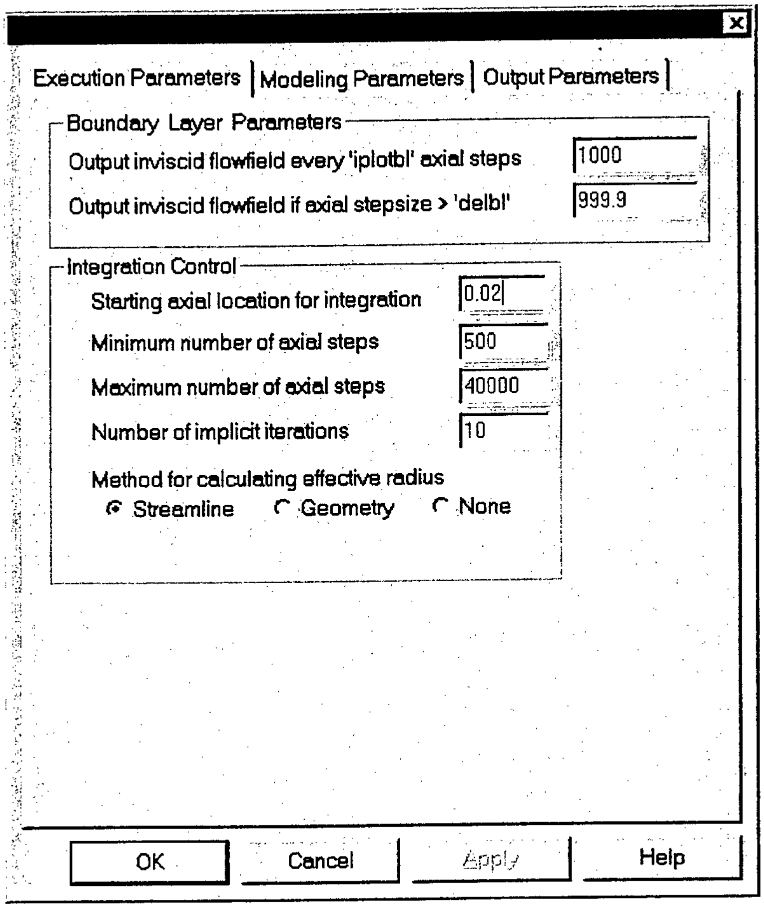
A.3.3 SEPARATION MODELING

This section is used to specify separation-modeling parameters, all of which are described in detail in the Separation Modeling section. Separation modeling is not used for the tutorial, and therefore, this section may be omitted.

A.3.4 BOUNDARY LAYER DATA

If an approximate viscous solution is desired then the following values must be specified. An approximate viscous solution is obtained by solving the integral form of the boundary layer equations. The integral boundary layer solution employed requires the inviscid solution from the Zeus Euler solver as a boundary condition. An in-depth description of all of the parameters shown below are described in the Boundary Layer Parameters section.

- Click on 'Inputs' and then select 'Boundary Layer Data'.
- Enter the values shown in the figure below and click on the 'Modeling Parameters' tab.



Boundary Layer Parameters

Output inviscid flowfield every 'iplotbl' axial steps: 1000

Output inviscid flowfield if axial stepsize > 'delbl': 999.9

Integration Control

Starting axial location for integration: 0.02

Minimum number of axial steps: 500

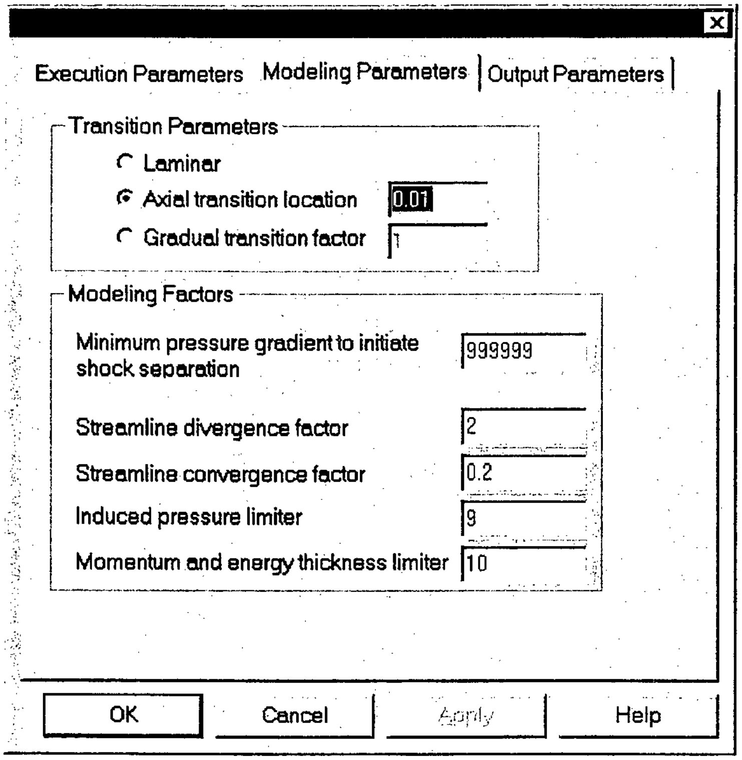
Maximum number of axial steps: 40000

Number of implicit iterations: 10

Method for calculating effective radius: ● Streamline ○ Geometry ○ None

FIGURE A-11. SAMPLE INPUTS FOR BOUNDARY LAYER EXECUTION PARAMETERS

- Enter the values shown in the figure below and click on 'OK'



Transition Parameters

○ Laminar
● Axial transition location 0.01
○ Gradual transition factor 1

Modeling Factors

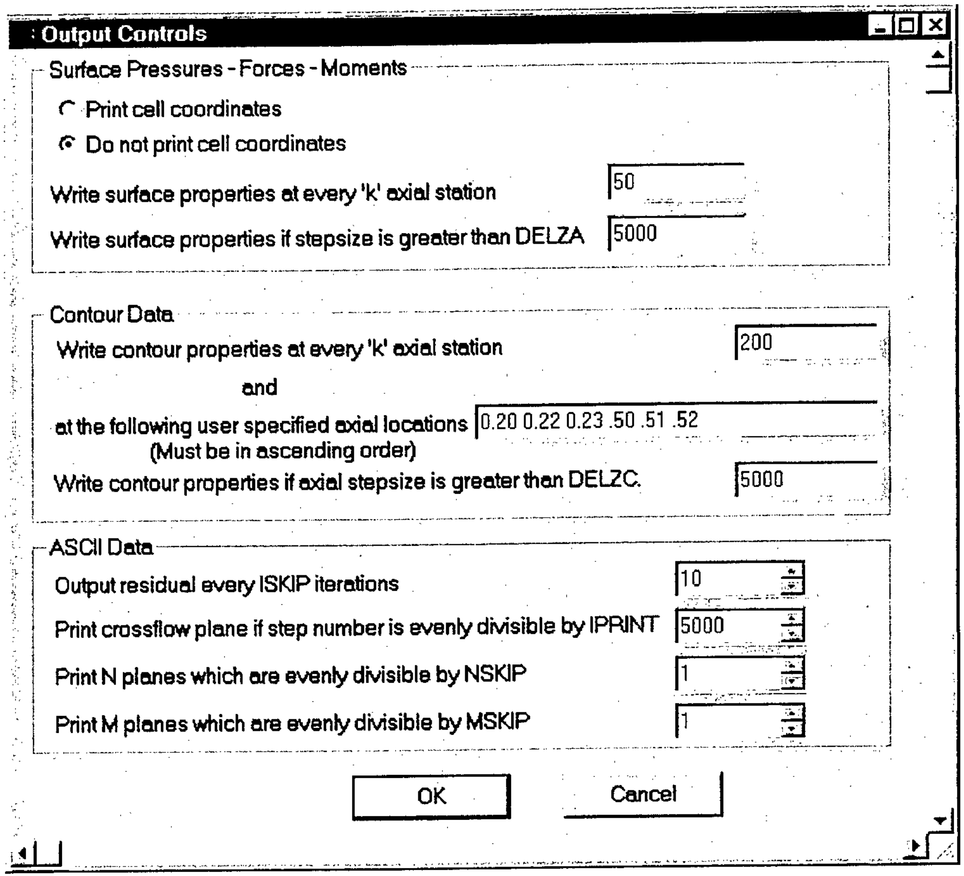
Minimum pressure gradient to initiate shock separation 999999
Streamline divergence factor 2
Streamline convergence factor 0.2
Induced pressure limiter 9
Momentum and energy thickness limiter 10

FIGURE A-12. SAMPLE INPUTS FOR BOUNDARY LAYER MODELING PARAMETERS

A.3.5 OUTPUT CONTROLS

This section is used to specify what type of output is generated, as well as the frequency of the output (with respect to axial integration steps). Each of the parameters shown below are described in detail in the Output Controls section.

- Click on 'Inputs' and then select 'Output Control'
- Enter all of the values shown in the figure below and click 'OK'.



Surface Pressures - Forces - Moments

○ Print cell coordinates

● Do not print cell coordinates

Write surface properties at every 'k' axial station 50

Write surface properties if stepsize is greater than DELZA 5000

Contour Data

Write contour properties at every 'k' axial station 200

and

at the following user specified axial locations 0.20 0.22 0.23 .50 .51 .52
(Must be in ascending order)

Write contour properties if axial stepsize is greater than DELZC. 5000

ASCII Data

Output residual every ISKIP iterations 10

Print crossflow plane if step number is evenly divisible by IPRINT 5000

Print N planes which are evenly divisible by NSKIP 1

Print M planes which are evenly divisible by MSKIP 1

FIGURE A-13. SAMPLE INPUTS FOR OUTPUT CONTROL SECTION

A.4 EXECUTION

RUNNING A CASE

This section is used to select which cases are evaluated as well as what type of solution is performed.

- Click 'Execution'.
- Click on the desired run.

Note: Multiple runs may be selected in a number of ways. The first is to hold down the 'Ctrl' key and click on each of the desired runs. Alternatively, you can click on the first desired run, hold down the 'shift' key, and then click on the last run to select all cases in between. Finally, you can simply click on the 'select All' button to run all cases.

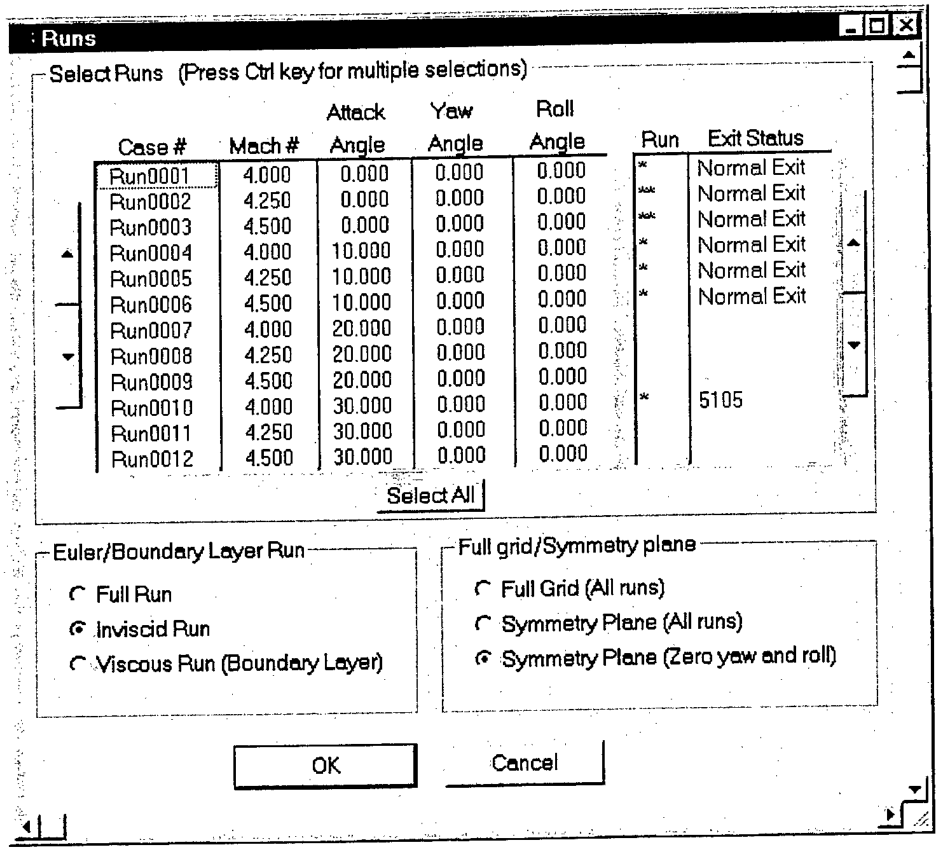
- Select the type of run desired.

Note: If 'Full Run' is selected the code will execute the Inviscid Euler solver, write out the flowfield at the specified axial intervals, and then execute the boundary layer solver to get an approximate viscous solution. If 'Inviscid Run' is selected, the code will execute the Inviscid Euler solver and then terminate. No viscous approximation will be calculated. If a viscous calculation is desired at later time, simply click on the 'Viscous Run' button, and select the desired case. Only those runs that have been solved with the inviscid solver will be displayed when the 'Viscous Run' button is selected. This is because the viscous approximation requires the inviscid flowfield as a boundary condition.

- Specify whether a symmetry plane exists.

Note: If you know a priori that the geometry is non-symmetric about the pitch plane, you must select 'Full Grid (All Runs)'. However, if the geometry is symmetric about the pitch plane choose either 'symmetry Plane (All Runs)' if you will not be running any asymmetric cases, or alternatively, choose 'symmetry Plane (Zero Roll and Yaw)' to run a symmetry plane only for those cases with zero yaw or roll angles. Running with a symmetry plane will cut the computational time approximately in half.

- Click on 'OK' to perform the desired calculations.
- After the runs have completed, you can go back to the 'Execution' section to check on the termination status for each run. If the 'Completed' column contains one '*', then the inviscid solution has completed. If it contains '**' then both the inviscid and viscous solutions have been completed. In addition, the 'Exit Status' column should contain 'Normal Exit' if no problems were encountered. If the code did not exit normally then an exit code will be provided. The exit codes, along with an explanation of each, can be found in the Exit Codes section.



Select Runs (Press Ctrl key for multiple selections)

Case #	Mach #	Attack Angle	Yaw Angle	Roll Angle	Run	Exit Status
Run0001	4.000	0.000	0.000	0.000	*	Normal Exit
Run0002	4.250	0.000	0.000	0.000	**	Normal Exit
Run0003	4.500	0.000	0.000	0.000	**	Normal Exit
Run0004	4.000	10.000	0.000	0.000	*	Normal Exit
Run0005	4.250	10.000	0.000	0.000	*	Normal Exit
Run0006	4.500	10.000	0.000	0.000	*	Normal Exit
Run0007	4.000	20.000	0.000	0.000		
Run0008	4.250	20.000	0.000	0.000		
Run0009	4.500	20.000	0.000	0.000		
Run0010	4.000	30.000	0.000	0.000	*	5105
Run0011	4.250	30.000	0.000	0.000		
Run0012	4.500	30.000	0.000	0.000		

Euler/Boundary Layer Run
- ○ Full Run
- ● Inviscid Run
- ○ Viscous Run (Boundary Layer)

Full grid/Symmetry plane
- ○ Full Grid (All runs)
- ○ Symmetry Plane (All runs)
- ● Symmetry Plane (Zero yaw and roll)

FIGURE A-14. SAMPLE SCREEN SHOT OF EXECUTION CONTROL PARAMETERS

A.5 POST-PROCESSING

OVERVIEW

The post-processing section of the tutorial is divided into three sections. To execute the plotting functions of the post-processor, a current version of the Tecplot[9] plotting software must be installed on the user's machine. If this software is not installed, the user may still examine the ASCII output data generated by Zeus^{++}.

ASCII Data	Examine an ASCII data file of the forces/moments acting on the body as well as the centers of pressure.
Single Run Plots	Line plots of the surface pressures, forces, and moments. Vector and streamline plots. Contour plots of a variety of flowfield parameters (Mach number, density, temperature, ...etc).
Multi-Run Plots	Comparative line plots of forces/moments versus a variety of independent variables (Mach number, angle of attack, yaw angle, and roll orientation).

A.5.1 ASCII FORCES & MOMENTS

This section is used to examine the ASCII output data from the Zeus^{++} runs. The data includes the forces, moments, and centers of pressure for both the entire geometry as well as each individual zone edge. An in-depth explanation of the ASCII data can be found in the ASCII Forces & Moments section of the users' manual.

- Click on 'Post-Processor' and then select 'ASCII Forces & Moments'
- Select 'Totals' to display the total forces/moments acting over the entire geometry. Alternatively, 'Edge Values' could be selected to display the individual components of the forces/moments that constitute the total.
- Microsoft WordPad will open and display the ASCII (forcesmoments.txt) version of the output data. The data can either be printed or saved to another file using this program. When finished examining the data, close the WordPad program to return to Zeus^{++}.

A.5.2 SINGLE-RUN PLOTS

This section is used to generate graphs (both line and contour) of the forces, moments, centers of pressure, and a variety of flowfield parameters (density, Mach number, pressure, etc...). Comparison plots of the various runs are generated in the Multi-Runs section.

- Click on 'Post-Processor' and then select 'single Run Plots'.
- Select 'Plot Surface Pressure'. A window will then be displayed which lists all completed runs. Select the desired run and click on 'OK' to generate the graph. A detailed description of the data in the surface pressure plot can be found in the users' manual in the Surface Pressure section.
- Exit the plotting package to return to Zeus^{++}.
- Click on 'Post-Processor' and then select 'single Run Plots'.
- Select 'Plot Forces & Moments' and then choose either 'Totals' or 'Edge Values' to generate the corresponding graph. After the completed runs are displayed, select the desired run and click on 'OK' to generate the graph. A description of the data in the graph is provided in the users' manual in the Forces & section.

- Exit the plotting package to return to Zeus^{++}.
- Click on 'Post-Processor' and then select 'single Run Plots'.
- Select 'Contour Plots' and then 'Choose Plot Attributes'. Enter the desired dimensionalization parameters for the contour plots, select the desired variables, and click on 'OK'. The above parameters are explained in detail in the Contour/Vector section of the user's manual.
- Return to the 'Contour Plots' menu, however, this time click on 'Plot Data' to generate the contour graph. After the completed runs are displayed, select the desired run and click on 'OK' to generate the graph.
- Exit the plotting package to return to Zeus^{++}.

A.5.3 MULTI-RUN PLOTS

This section is used to make comparison plots of different runs. The options include comparisons of all forces, moments, and centers of pressure with a choice of dependant variables (Mach number, angle of attack/yaw/roll).

- Click on 'Post-Processor' and then select 'Multi-Run Plots'.
- Select'Choose Plot Attributes' and specify the values to be plotted as well as the independent variable. Refer to the Multi-Plot run section of the user's manual.
- Return to the 'Multi-Run Plots' menu, however, this time click on 'Plot Data' to generate the comparison plot for all executed runs. If, during the tutorial, only a single run was executed, then the line plot displayed will be a point. In order to make comparative plots of different runs, return to the execution section, and run additional cases.
- Exit the plotting package to return to Zeus^{++}.

DISTRIBUTION

Copies

DOD ACTIVITIES (CONUS)

ATTN CODE 35 (ZIMET) 1
CODE 351 (SIEGEL) 1
CODE 351 (CHEW) 1
CODE 332FD (LEKOUDIS) 1
CHIEF OF NAVAL RESEARCH
BALLSTON CENTRE TOWER ONE
800 NORTH QUINCY ST
ARLINGTON VA 22217-5660

ATTN CODE 474T6OD (LOFTUS) 1
CODE 4732HOD (SMITH) 1
CODE 473COOD (PORTER) 1
CODE 47311OD (HOUSH) 1
CODE 47311OD (GLEASON) 1
CODE 4722EOD (JETER) 1
TECHNICAL LIBRARY 1
COMMANDER
NAVAL AIR WARFARE CENTER
1 ADMINISTRATION CIRCLE
CHINA LAKE CA 93555-6001

ATTN TECHNICAL LIBRARY 1
G RUDACILLE PMS 38012 7 1
COMMANDER
NAVAL SEA SYSTEMS COMMAND
2531 JEFFERSON DAVIS HWY
ARLINGTON VA 22242-5160

ATTN TECHNICAL LIBRARY 1
COMMANDER
NAVAL AIR SYSTEMS COMMAND
47122 LILJENCRANTZ ROAD UNIT 7
PATUXENT RIVER MD 20670-5440

ATTN C KLEIN 1
TECHNICAL LIBRARY 1
COMMANDER
NAVAL AIR WARFARE CENTER
WEAPONS DIVISION
521 9TH ST
POINT MUGU CA 93042-5001

Copies

ATTN T C TAI 1
M J MALIA 1
TECHNICAL LIBRARY 1
COMMANDER
NSWC CARDEROCK DIVISION
WASHINGTON DC 20034

ATTN R M HOWARD 1
TECHNICAL LIBRARY 1
SUPERINTENDENT
NAVAL POSTGRADUATE SCHOOL
1 UNIVERSITY CIRCLE
MONTEREY CA 93943-5001

ATTN HEAD WEAPONS DEPT 1
HEAD SCIENCE DEPT 1
SUPERINTENDENT
UNITED STATES NAVAL ACADEMY
121 BLAKE RD
ANNAPOLIS MD 21402-5000

ATTN DIAG DT 4T (PAUL MURAD) 2
DIRECTOR
DEFENSE INTELLIGENCE AGENCY
WASHINGTON DC 20301

ATTN BRENT WAGGONER 1
CODE 4072 BLDG 2540
NAVAL WEAPONS SUPPORT CENTER
CRANE IN 47522-5000

ATTN CODE 5252P (KRAUSE) 1
TECHNICAL LIBRARY 1
COMMANDER
INDIAN HEAD DIVISION
NAVAL SURFACE WARFARE CENTER
101 STRAUSS AVE
INDIAN HEAD MD 20640-5035

ATTN TECHNICAL LIBRARY 1
COMMANDING GENERAL
MARINE CORPS COMBAT
DEVELOPMENT COMMAND
2048 SOUTH ST
QUANTICO VA 22134-5129

DISTRIBUTION (Continued)

	Copies
ATTN E SEARS	1
L E LIJEWSKI	1
C COTTRELL	1
TECHNICAL LIBRARY	1
AFATL (ADLRA) (DLGC)	1
EGLIN AFB FL 32542-5000	
ATTN TECHNICAL LIBRARY	1
USAF ACADEMY	
COLORADO SPRINGS CO 80912	
ATTN TECHNICAL LIBRARY	1
ADVANCED RESEARCH PROJECTS AGENCY	
DEPARTMENT OF DEFENSE	
WASHINGTON DC 20305	
ATTN B BLAKE (BLD 146)	1
J JENKINS (BLD 146)	1
TECHNICAL LIBRARY	1
COMMANDING OFFICER	
AFSC	
2210 8TH STREET	
WRIGHT PATTERSON AFB OH 45433	
ATTN EDWARD JENKINS	1
NAIC TANW	
HQ NAIC TANW	
4115 HEBBLE CREEK ROAD SUITE 28	
WPAFB OH 45433-5623	
ATTN J USSELTON	1
W B BAKER JR	1
TECHNICAL LIBRARY	1
ARNOLD ENGINEERING DEVELOPMENT CENTER USAF	
TULLAHOMA TN 37389	
ATTN H HUDGINS	1
G FRIEDMAN	1
TECHNICAL LIBRARY	1
COMMANDING GENERAL	
ARRADCOM PICATINNY ARSENAL	
DOVER NJ 07801	

	Copies
ATTN R PUHALLA JR	1
W STUREK	1
C NIETUBICZ	1
A MIKHAIL	1
P PLOSTINS	1
TECHNICAL LIBRARY	1
COMMANDING GENERAL	
BALLISTIC RESEARCH LABORATORY	
ABERDEEN PROVING GROUND	
ABERDEEN MD 21005-5066	
ATTN CODE TNC (BLACKLEDGE)	1
RICH MATLOCK	1
DIRECTOR	
INTERCEPTOR TECHNOLOGY	
BALLISTIC MISSILE DEFENSE OFFICE	
THE PENTAGON	
WASHINGTON DC 20350	
ATTN SFAE SD ASP	1
SFAE SD HED	1
DEPUTY COMMANDER	
US ARMY STRATEGIC DEFENSE COMMAND	
P O BOX 1500	
HUNTSVILLE AL 35807-3801	
ATTN D WASHINGTON	1
W WALKER	1
R KRETZSCHMAR	1
D FERGUSON JR	1
COMMAND GENERAL	
US ARMY MISSILE COMMAND	
AMSMI RD SS AT	
REDSTONE ARSENAL AL 35898-5252	
DEFENSE TECHNICAL INFORMATION CENTER	
8725 JOHN J KINGMAN ROAD	
SUITE 0944	
FORT BELVOIR VA 22060-6218	2
DIRECTOR	
DEFENSE PRINTING SERVICE	
BLDG 176 WASHINGTON NAVY YARD	
901 M ST E	
WASHINGTON DC 20474-5087	

DISTRIBUTION (Continued)

Copies

ATTN CODE A76
TECHNICAL LIBRARY 1
COMMANDING OFFICER
CSSDD NSWC
6703 W HIGHWAY 98
PANAMA CITY FL 32407-7001

ATTN DR P WEINACHT 1
AERODYNAMICS BRANCH
PROPULSION AND FLIGHT DIV WTD
AMSRL WT PB
US ARMY RESEARCH LAB ABERDEEN
PROVING GROUND MD 21005-5066

ATTN GREGG ABATE 1
US AIR FORCE
WRIGHT LABORATORY
WL MNAA
101 W EGLIN BLVD STE 219
EGLIN AFB FL 32542-5000

ATTN JOHN GRAU 1
US ARMY ARDEC
COMMANDER US ARMY ARDEC
AMSTA AR AET A BLDG 3342
PICATINNY ARSENAL NJ 07806-5000

ATTN FRANK MACDONALD 1
NAWC CHINA LAKE
COMMANDER
CODE 473 20D
NAVAIRWARCENNSDNDIV
CHINA LAKE CA 93555

ATTN MARK LAMBERT 1
NAWC
CODE 4732HOD
CHINA LAKE CA 93555

ATTN MICHAEL MUSACHIO 1
DIRECTOR
OFFICE OF NAVAL INTELLIGENCE
4251 SUITLAND ROAD (ONI 2321)
WASHINGTON DC 20395

Copies

ATTN DR ALAN NICHOLSON MSC 5B 1
DEFENSE INTELLIGENCE AGENCY
MISSILE AND SPACE INTELLIGENCE CTR
REDSTONE ARSENAL AL 35898-5500

ATTN EDWARD HERBERT 1
US ARMY MISSILE COMMAND
AMSMI RD MG GA
BLDG 5400 ROOM 250
REDSTONE ARSENAL AL 35898

ATTN PAUL KOLODZIEJ 1
NASA AMES RESEARCH CENTER
MS 234 1
MOFFETT FIELD CA 94035

ATTN LCDR T HARTLINE USNR R 1
NR ONI 2109 NAVAL RESERVE UNIT
112 CRESTVIEW CIRCLE
MADISON AL 35758

ATTN CODE 4732HOD DAVID HALL 1
PROPULSION PERFORMANCE OFFICE
NAVAL AIR WARFARE CTR WEAPONS DIV
1 ADMINISTRATIVE CIR
CHINA LAKE CA 93555-6001

ATTN DONALD SHEREDA 1
WL FIMA BLDG 450
2645 FIFTH ST STE 30
WRIGHT PATTERSON AFB OH 45433-7936

ATTN ROBERT VAN DYKEN 473110D 1
COMMANDER
NAWC
CHINA LAKE CA 93555

BMDO AQS
1725 JEFFERSON DAVIS HWY STE 809
ARLINGTON VA 22202 1

DISTRIBUTION (Continued)

Copies

ATTN JEFFREY RANDORF 1
US ARMY SPACE AND STRATEGIC
DEFENSE COMMAND
P O BOX 1500 CSSD-BC-SS
106 WYNN DRIVE
HUNTSVILLE AL 35807-3801

ATTN TECHNICAL LIBRARY 1
NEILSON ENGINEERING AND
RESEARCH INC
526 CLYDE AVENUE
MOUNTAIN VIEW CA 95043

ATTN TECHNICAL LIBRARY 1
MCDONNEL DOUGLAS
ASTRONAUTICS CO (WEST)
5301 BOLSA AVENUE
HUNTINGTON BEACH CA 92647

ATTN TECHNICAL LIBRARY 1
LOCKHEED MISSILES AND SPACE CO INC
PO BOX 1103
HUNTSVILLE AL 35807

NON-DOD ACTIVITIES (CONUS)

NICHOLS RESEARCH CORPORATION
MS 912
P O BOX 400002
4040 S MEMORIAL PKWY
HUNTSVILLE AL 35815-1502 1

THE CNA CORPORATION
P O BOX 16268
ALEXANDRIA VA 22302-0268 1

GIDEP OPERATIONS OFFICE
CORONA CA 91720 1

ATTN TECHNICAL LIBRARY 1
NASA AMES RESEARCH CENTER
MOFFETT CA 94035-1099

ATTN C SCOTT 1
D CURRY 1
NASA JOHNSON SPACE CENTER
HOUSTON TX 77058

Copies

ATTN TECHNICAL LIBRARY 1
NASA
WASHINGTON DC 20546

ATTN B HENDERSON 1
D MILLER 1
J ALLEN 1
F WILCOX 1
TECHNICAL LIBRARY 2
NASA LANGLEY RESEARCH CENTER
HAMPTON VA 23365

ATTN D G MILLER (L 219) 1
TECHNICAL LIBRARY 1
LAWRENCE LIVERMORE NATIONAL
LABORATORY
EARTH SCIENCES DIVISION
UNIVERSITY OF CALIFORNIA
P O BOX 808
LIVERMORE CA 94551

ATTN W RUTLEDGE (1635) 1
R LAFARGE 1
R EISLER 1
TECHNICAL LIBRARY 1
SANDIA NATIONAL LABORATORY
P O BOX 5800
ALBUQUERQUE NM 87185-5800

ATTN WALT GUTIERREZ 1
SANDIA NATIONAL LABORATORIES
MAIL STOP 0825
P O BOX 5800
ALBUQUERQUE NM 87185-0825

ATTN ASSISTANT DEFENSE
COOPERATION ATTACHE 1
EMBASSY OF SPAIN
WASHINGTON DC 20016

DE/AVT
DEFENSE EQUIPMENT STAFF
BRITISH EMBASSY
3100 MASSACHUSETTS AVE NW
WASHINGTON DC 20008-3688 1

DISTRIBUTION (Continued)

Copies

ATTN ASO LO IS 1
ISRAEL AIR FORCE
LIAISON OFFICER
700 ROBBINS AVE
PHILADELPHIA PA 19111

ATTN GERMAN MILITARY REP US OA 1
GMR TRAFFIC AND TRANSPORTATION
DIVISION
10 SERVICES ROAD
DULLES INTERNATIONAL AP
WASHINGTON DC 20041

ATTN PROF F R DEJARNETTE 1
NORTH CAROLINA STATE UNIVERSITY
DEPT OF MECHANICAL AND
AEROSPACE ENGINEERING
BOX 7921
RALEIGH NC 27695

ATTN PROF J A SCHETZ 1
VIRGINIA POLYTECHNIC AND STATE
UNIVERSITY
DEPT OF AEROSPACE ENGINEERING
BLACKSBURG VA 24060

ATTN J M WU 1
C BALASUBRAMAYAN 1
TECHNICAL LIBRARY 1
THE UNIVERSITY OF TENNESSEE
SPACE INSTITUTE
TULLAHOMA TN 37388

ATTN R NELSON 1
TECHNICAL LIBRARY 1
UNIVERSITY OF NOTRE DAME
DEPT OF AEROSPACE AND
MECHANICAL ENGINEERING
BOX 537
NOTRE DAME IN 46556

ATTN PROF F NELSON 1
DEPT OF MECH AND AERO ENG
UNIVERSITY OF MISSOURI ROLLA
ROLLA MO 65401

Copies

ATTN ROBERT ENGLAR 1
GEORGIA TECH RESEARCH INSTITUTE
AEROSPACE SCIENCE AND
TECHNOLOGY LAB
ATLANTA GA 30332

ATTN E LUCERO 1
D FROSTBUTTER 1
L PERINI 1
TECHNICAL LIBRARY 1
APPLIED PHYSICS LABORATORY
JOHNS HOPKINS UNIVERSITY
JOHNS HOPKINS ROAD
LAUREL MD 20723-6099

ATTN B BROOKS 1
R STANCIL 1
R ELKINS 1
LORAL VOUGHT SYSTEMS
P O BOX 650003
M S EM 55
DALLAS TX 75265-0003

ATTN PROF J D ANDERSON 1
DEPT OF AEROSPACE ENGINEERING
UNIVERSITY OF MARYLAND
COLLEGE PARK MD 20742

ATTN TECHNICAL LIBRARY 1
HUGHES MISSILE SYSTEMS COMPANY
P O BOX 11337 BLDG 802 MS A1
OLD NOGALES HWY
TUCSON AZ 83734-1337

ATTN M DILLENIUS 1
NIELSEN ENGINEERING AND
RESEARCH INC
526 CLYDE AVE
MOUNTAIN VIEW CA 95043

ATTN J XERIKOS 1
N CAMPBELL 1
TECHNICAL LIBRARY 1
MCDONNEL DOUGLAS
ASTRONAUTICS CO (WEST)
5301 BOLSA AVE
HUNTINGTON BEACH CA 92647

DISTRIBUTION (Continued)

Copies

ATTN J WILLIAMS 1
S VUKELICH 1
J FIVEL 1
R GERBSCH (CODE 1111041) 1
TECHNICAL LIBRARY 1
MCDONNELL DOUGLAS
ASTRONAUTICS CO (EAST)
BOX 516
ST LOUIS MO 63166-0516

ATTN TECHNICAL LIBRARY 1
UNITED TECHNOLOGIES
NORDEN SYSTEMS
NORWALK CT 06856

ATTN T LUNDY 1
D ANDREWS 1
TECHNICAL LIBRARY 1
LOCKHEED MISSILES AND SPACE CO INC
P O BOX 1103
HUNTSVILLE AL 35807

ATTN W CHRISTENSON 1
D WARNER 1
ALLIANT TECHSYSTEMS INC
600 SECOND ST NE
HOPKINS MN 55343

ATTN TECHNICAL LIBRARY 1
B SALEMI 1
J BOUDREAU 1
RAYTHEON COMPANY
MISSILE SYSTEMS DIVISION
P O BOX 1201
TEWKSBURY MA 01876-0901

ATTN LLOYD PRATT 1
AEROJET TACTICAL SYSTEMS CO
P O BOX 13400
SACRAMENTO CA 95813

ATTN JOSEPH ANDRZEJEWSKI 1
MEVATEC CORP
1525 PERIMETER PARKWAY
SUITE 500
HUNTSVILLE AL 35806

Copies

ATTN DR G S SCHMIDT 1
LORAL DEFENSE SYSTEMS
1210 MASSILLON ROAD
AKRON OH 44315-0001

ATTN W NORDGREN 721 1
GOULD INC OSD
18901 EUCLID AVE
CLEVELAND OH 44117

ATTN TECH LIBRARY 1
AEROJET ELECTRONIC SYSTEMS
P O BOX 296 III
AZUSA CA 91702

ATTN P REDING 1
G CHRUSCIEL 1
TECHNICAL LIBRARY 1
LOCKHEED MISSILES AND SPACE CO INC
P O BOX 3504
SUNNYVALE CA 94088

ATTN K C LEE 1
AEROTHERM CORP
580 CLYDE AVE
MOUNTAIN VIEW CA 94043

ATTN TECH LIBRARY 1
FMC NAVAL SYSTEMS DIV
4800 E RIVER ROAD
MINNEAPOLIS MN 55421-1402

ATTN DORIA GLADSTONE 1
BATTELLE MEMORIAL INSTITUTE
COLUMBUS DIVISION
505 KING AVE
COLUMBUS OH 43201-2693

ATTN JAMES SORENSON 1
VINCENT ALLEN 1
ORBITAL SCIENCES
3380 SOUTH PRICE ROAD
CHANDLER AZ 85248

DISTRIBUTION (Continued)

Copies

ATTN J FORKOIS 1
KAMAN SCIENCES CORP
1500 GARDEN OF THE GODS ROAD
P O BOX 7463
COLORADO SPRINGS CO 80933

ATTN RON EFROMSON 1
MIT LINCOLN LABORATORY
244 WOOD STREET
LEXINGTON MA 02173-0073

ATTN DJ GIESE 1
MAIL STOP 4C 61
BOEING DEFENSE AND SPACE GROUP
PO BOX 3999
SEATTLE WA 98124-2499

ATTN BRIAN WALKUP 1
ALLEGHENY BALLISTICS LAB
210 STATE ROUTE 956 WV01-13
ROCKET CENTER WV 26726-3548

ATTN DR T LIN 1
TRW ELECTRONICS AND DEFENSE SECTOR
BLDG 527/RM 706
P O BOX 1310
SAN BERNADINO CA 92402

ATTN G VINCENT 1
SPARTA INC
4901 CORPORATE DR
HUNTSVILLE AL 35805

ATTN D P FORSMO 1
TECHNICAL LIBRARY 1
RAYTHEON COMPANY
MISSILE SYSTEMS DIVISION
HARTWELL RD
BEDFORD MA 01730-2498

ATTN M S MILLER 1
N R WALKER 1
DYNETICS INC
P O DRAWER B
HUNTSVILLE AL 35814-5050

Copies

ATTN H A MCELROY 1
GENERAL DEFENSE CORP
P O BOX 127
RED LION PA 17356

ATTN ENGINEERING LIBRARY 1
ARMAMENT SYSTEMS DEPT
GENERAL ELECTRIC CO
BURLINGTON VT 05401

ATTN TECHNICAL LIBRARY 1
OAYNE AERONAUTICAL
2701 HARBOR DRIVE
SAN DIEGO CA 92138

ATTN BRIAN EST 1
BOEING ST LOUIS
P O BOX 516
ST LOUIS MO 63166-0516

ATTN WILLIAM FACINELLI 1
ALLIED SIGNAL
P O BOX 22200
MS 1207 3B
TEMPE AZ 85285

ATTN DR T P SHIVANANDA 1
TRW BMD
P O BOX 1310
SAN BERNADINO CA 92402-1313

ATTN T R PEPITONE 1
AEROSPACE TECHNOLOGY INC
P O BOX 1809
DAHLGREN VA 22448

ATTN ERIC MOORE 1
MAIL STOP MER 24 1281
LOCKHEED SANDERS
P O BOX 868
NASHUA NH 03061

ATTN DR BRIAN LANDRUM 1
RI BLDG E33
PROPULSION RESEARCH CENTER
UNIVERSITY OF ALABAMA
HUNTSVILLE AL 35899

DISTRIBUTION (Continued)

Copies

ATTN BRUCE NORTON 1
MAIL STOP BL 1
RAYTHEON
100 VANCE TANK RD
BRISTOL TN 37620

ARROW TECH ASSOCIATES INC
1233 SHELBURNE ROAD D8
SO BURLINGTON VT 05403

ATTN JUAN AMENABAR 1
SAIC
4001 NORTH FAIRFAX DRIVE STE 800
ARLINGTON VA 22209

ATTN TECHNICAL LIBRARY 1
TELEDYNE RYAN AERONAUTICAL
2701 HARBOR DRIVE
SAN DIEGO CA 92138

ATTN DR KIRIT PATEL 1
SVERDRUP TECHNOLOGY INC
TEAS GROUP
BLDG 260 P O BOX 1935
EGLIN AFB FL 32542

ATTN FRANK LANGHAM 1
MICRO CRAFT TECHNOLOGY
740 4TH ST
MS 6001
ARNOLD AFB TN 37389

ATTN LAURA AYERS 1
DELTA RESEARCH INC
315 WYNN DRIVE
SUITE 1
HUNTSVILLE AL 35805

ATTN BRIAN BENNETT 1
MCDONNELL DOUGLAS
MC 064 2905
P O BOX 516
ST LOUIS MO 63166-0516

Copies

ATTN THOMAS FARISS 1
LOCKHEED SANDERS
P O BOX 868
MER24 1206
NASHUA NH 03061-0868

ATTN JEFFREY HUTH 1
KAMAN SCIENCES CORPORATION
2560 HUNTINGTON AVE
ALEXANDRIA VA 22303

ATTN WILLIAM JOLLY 1
KAMAN SCIENCES
600 BLVD SOUTH SUITE 208
HUNTSVILLE AL 35802

ATTN STEPHEN MALLETTE 1
KBM ENTERPRISES
15980 CHANEY THOMPSON RD
HUNTSVILLE AL 35803

ATTN DONALD MOORE 1
NICHOLS RESEARCH CORPORATION
4040 SOUTH MEMORIAL PARKWAY
P O BOX 400002
MS 920C
HUNTSVILLE AL 35815-1502

ATTN NANCY SWINFORD 1
LOCKHEED MISSILES & SPACE CO
P O BOX 3504
ORG E5-40 BLDG 1575E
SUNNYVALE CA 94088-3504

ATTN DAVID RESSLER 1
TRW BALLISTIC MISSILES DIV
MS 953 2420
P O BOX 1310
SAN BERNARDINO CA 92402

ATTN MARK SWENSON 1
ALLIANT TECHSYSTEMS
MN11 262B
600 SECOND STREET NE
HOPKINS MN 55343

DISTRIBUTION (Continued)

Copies

ATTN LEROY M HAIR 1
COLEMAN RESEARCH CORP
6820 MOQUIN DRIVE
HUNTSVILLE AL 35806

ATTN SCOTT ALLEN 1
ALLEN AERO RESEARCH
431 E SUNNY HILLS RD
FULLERTON CA 92635

ATTN DARRYL HALL 1
SAIC
997 OLD EAGLE SCHOOL RD
SUITE 215
WAYNE PA 19087-1803

ATTN PETER ALEXANDER 1
MCDONNELL DOUGLAS AEROSPACE
689 DISCOVERY DRIVE
MS 11A1
HUNTSVILLE AL 35806

ATTN SAMUEL HICKS III 1
TEXAS INSTRUMENTS
6600 CHASE OAKS BLVD
MS 8490
PLANO TX 75086

ATTN BARRY LINDBLOM 1
ALLIANT DEFENSE ELECTRONICS
SYSTEMS INC
P O BOX 4648
CLEARWATER FL 34618

ATTN DR SHIN CHEN 1
THE AEROSPACE CORP
M4 967
P O BOX 92957
LOS ANGELES CA 90009

ATTN ROBERT ACEBAL 1
SAIC
1225 JOHNSON FERRY RD
SUITE 100
MARIETTA GA 30068

Copies

ATTN EUGENE HART 1
SYSTEM PLANNING CORP
1000 WILSON BLVD
ARLINGTON VA 22209

ATTN ELAINE POLHEMUS 1
ROCKWELL AUTONETICS & MISSILE
SYSTEMS DIVISION
D611 DL23
1800 SATELLITE BLVD
DULUTH GA 30136

ATTN MICHAEL GLENN 1
TASC
1992 LEWIS TURNER BLVD
FT WALTON BEACH FL 32547

ADAPTIVE RESEARCH
4960 CORPORATE DRIVE
SUITE 100 A
HUNTSVILLE AL 35805-6229 1

ATTN STEVEN MARTIN 1
SYSTEMS ENGINEERING GROUP INC
9841 BROKEN LAND PARKWAY
SUITE 214
COLUMBIA MD 21046-1120

ATTN C W GIBKE 1
LOCKHEED MARTIN VOUGHT SYSTEMS
MS SP 72
P O BOX 650003
DALLAS TX 75265-0003

ATTN CHRIS HUGHES 1
EDO GOVERNMENT SYSTEMS DIV
14 04 111TH ST
COLLEGE POINT NY 11356

ATTN DANIEL LESIEUTRE 1
NIELSEN ENGINEERING & RES INC
526 CLYDE AVENUE
MOUNTAIN VIEW CA 94043-2212

DISTRIBUTION (Continued)

Copies

ATTN CARL HILL 1
FRANCIS PRIOLO 1
STANDARD MISSILE COMPANY LLC
1505 FARM CREDIT DRIVE
SUITE 600
MCLEAN VA 22102

ATTN JENNIE FOX 1
LOCKHEED MARTIN VOUGHT SYSTEMS
P O BOX 650003
MS EM 55
DALLAS TX 75265-0003

ATTN JOHN BURKHALTER 1
AUBURN UNIVERSITY
211 AEROSPACE ENGR BLDG
AUBURN UNIVERSITY AL 36849

ATTN DR MAX PLATZER 1
NAVAL POSTGRADUATE SCHOOL
DEPT OF AERONAUTICS &
ASTRONAUTICS
CODE AA PL
MONTEREY CA 93943

ATTN MIKE DANGELO 1
MIT LINCOLN LABORATORY
1745 JEFFERSON DAVIS HWY 1100
ARLINGTON VA 22202

ATTN RICHARD HAMMER 1
JOHNS HOPKINS APPLIED PHYSICS LAB
JOHNS HOPKINS ROAD
LAUREL MD 20723-6099

ATTN MAURICE TUCKER 1
BATTELLE HUNTSVILLE OPERATIONS
7501 S MEMORIAL PKWY STE 101
HUNTSVILLE AL 35802

ATTN STEVE MULLINS 1
SIMULATION AND ENGINEERING CO INC
8840 HWY 20 STE 200 N
MADISON AL 35758

Copies

ATTN ROBERT BRAENDLEIU 1
KAISER MARQUARDT
16555 SATICOY ST
VAN NUYS CA 91406-1739

ATTN THOMAS LOPEZ 1
COLEMAN RESEARCH CORP
990 EXPLORER BLVD
HUNTSVILLE AL 35806

ATTN LAWRENCE FINK 1
BOEING DEFENSE AND SPACE GROUP
P O BOX 3999 MS 82-23
SEATTLE WA 98124

ATTN ROY KLINE 1
KLINE ENGINEERING CO INC
27 FREDON GREENDELL RD
NEWTON NJ 07860-5213

ATTN THOMAS KLAUSE 1
TRW
P O BOX 80810
ALBUQUERQUE NM 87198

ATTN DAN PLATUS 1
THE AEROSPACE CORPORATION
P O BOX 92957
LOS ANGELES CA 90009

ATTN DR REX CHAMBERLAIN 1
TETRA RESEARCH CORPORATION
2610 SPICEWOOD TR
HUNTSVILLE AL 35811-2604

ATTN DR DANNY LIU 1
ZONA TECHNOLOGY INC
2651 W GUADALUPE RD SUITE B 228
MESA AZ 85202

ATTN PERRY PETERSEN 1
NORTHROP GRUMMAN CORP
DEPT 9B51 MAIL ZONE XA
8900 EAST WASHINGTON BLVD
PICO RIVERA CA 90660-3783

DISTRIBUTION (Continued)

Copies

ATTN DR JAMES HAUSER 1
AERO SPECTRA INC
2850 KENYON CIRCLE
P O BOX 3006
BOULDER CO 80307

ATTN DARRELL AUSHERMAN 1
TRW SPACE AND DEFENSE
ONE SPACE PARK
MAIL STATION R1-1062
REDONDO BEACH CA 90278-1071

ATTN JAY EBERSOHL 1
ADVATECH PACIFIC INC
2015 PARK AVENUE SUITE 8
REDLANDS CA 92373

ATTN EDWARD RAWLINSON 1
SY TECHNOLOGY INC
4900 UNIVERSITY SQUARE SUITE 8
HUNTSVILLE AL 35816
ATTN LAYNE COOK 1
UNIVERSAL SPACE LINES
8620 WOLFF CT SUITE 110
WESTMINSTER CO 80030

ATTN PAUL WILDE 1
ACTA INC
2790 SKYPARK DR SUITE 310
TORRANCE CA 90505-5345

ATTN DR MICHAEL HOLDEN 1
CALSPAN UB RESEARCH CENTER
P O BOX 400
BUFFALO NY 14225

ATTN RICHARD GRABOW 1
SPACE VECTOR CORP
17330 BROOKHURST ST SUITE 150
FOUNTAIN VALLEY CA 92708

ATTN BRENT APPLEBY 1
DRAPER LABORATORY
555 TECHNOLOGY SQ MS77
CAMBRIDGE MA 02139

Copies

ATTN JAMES JONES 1
SPARTA INC
1901 N FORT MYER DR SUITE 600
ARLINGTON VA 22209

ATTN SCOTT HOUSER 1
PHOENIX INTEGRATION
1872 PRATT DRIVE SUITE 1835
BLACKSBURG VA 24060

ATTN S ROM MURTY 1
TELEDYNE BROWN ENGINEERING
MS 200
300 SPARKMAN DRIVE
HUNTSVILLE AL 35807

ATTN STUART COULTER 1
SVERDRUP TECHNOLOGY
670 2ND ST MS4001
ARNOLD AIR FORCE BASE
TULLAHOMA TN 37389-4001

ATTN DR RICHARD HOWARD 1
NAVAL POSTGRADUATE SCHOOL
DEPT OF AERONAUTICS AND
ASTRONAUTICS
CODE AA HO NPS
MONTEREY CA 93943

ATTN J BRENT RUMINE 1
MIT LINCOLN LABORATORY
244 WOOD STREET
BUILDING S ROOM 52-327
LEXINGTON MA 02173-9185

NON-DOD ACTIVITIES (EX-CONUS)

ATTN LOUIS CHAN 1
INSTITUTE FOR AEROSPACE
RESEARCH
NATIONAL RESEARCH COUNSIL
MONTREAL RD
OTTAWA ONTARIO
CANADA K1A0R6

DISTRIBUTION (Continued)

Copies

ATTN H B ASLUND 1
SAAB MILITARY AIRCRAFT
581 88 LINKOEPING
SWEDEN

ATTN A BOOTH 1
BRITISH AEROSPACE DEFENCE LTD
MILITARY AIRCRAFT DIVISION
WARTON AERODROME WARTON PRESTON
LANCASHIRE PR4 1AX
UNITED KINGDOM

ATTN J EKEROOT 1
BOFORS MISSILES
691 80 KARLSKOGA
SWEDEN

ATTN CH FRANSSON 1
NATIONAL DEFENCE RESEARCH
ESTABLISHMENT
DEPT OF WEAPON SYSTEMS EFFECTS
AND PROTECTION
KARLAVAGEN 106B
172 90 SUNDBYBERG
SWEDEN

ATTN M HARPER BOURNE 1
DEFENCE RESEARCH AGENCY
Q134 BUILDING
RAE FARNBOROUGH
HAMPSHIRE QU14 6TD
UNITED KINGDOM

ATTN A H HASSELROT 1
FFA
P O BOX 11021
161 11 BROMMA
SWEDEN

ATTN B JONSSON 1
DEFENCE MATERIAL ADMINISTRATION
MISSILE TECHNOLOGY DIVISION
115 88 STOCKHOLM
SWEDEN

Copies

ATTN R CAYZAC 1
GIAT INDUSTRIES
7 ROUTE DE GUERCY
18023 BOURGES CEDEX
FRANCE

ATTN MAJ F DE COCK 1
ECOLE ROYALE MILITAIRE
30 AV DE LA RENAISSANCE
1040 BRUXELLES
BELGIUM

ATTN P LEZEAUD 1
DASSAULT AVIATION
78 QUAI MARCEL DASSAULT
92214 SAINT CLOUD
FRANCE

ATTN J LINDHOUT 1
N L R
ANTHONY FOKKERWEG 2
1059 CM AMSTERDAM
THE NETHERLANDS

ATTN A MICKELLIDES 1
GEC MARCONI
DEFENCE SYSTEMS LTD
THE GROVE WARREN LANE
STANMORE MIDDLESEX
UNITED KINGDOM

ATTN K MOELLER 1
BODENSEEWERK
GERAETETECHNIK GMBH
POSTFACH 10 11 55
88641 UBERLINGEN
GERMANY

ATTN G MOSS 1
ROYAL MILITARY COLLEGE
AEROMECHANICAL SYSTEMS GROUP
SHRIVENHAM SWINDON
WILTS SN6 8LA
UNITED KINGDOM

DISTRIBUTION (Continued)

Copies

ATTN RIBADEAU DUMAS 1
MATRA DEFENSE
37 AV LOUIS BREGUET
BP 1
78146 VELIZY VILLACOUBLAY CEDEX
FRANCE

ATTN R ROGERS 1
DEFENCE RESEARCH AGENCY
BLDG 37
TUNNEL SITE
CLAPHAM BEDS MK 41 6AE
UNITED KINGDOM

ATTN S SMITH 1
DEFENCE RESEARCH AGENCY
Q134 BUILDING
RAE FARNBOROUGH
HAMPSHIRE QU14 6TD
UNITED KINGDOM

ATTN J SOWA 1
SAAB MISSILES AB
581 88 LINKOPING
SWEDEN

ATTN D SPARROW 1
HUNTING ENGINEERING LTD
REDDINGS WOOD
AMPTHILL
BEDFORDSHIRE MK452HD
UNITED KINGDOM

ATTN P STUDER 1
DEFENCE TECHNOLOGY AND
PROCUREMENT AGENCY
SYSTEMS ANALYSIS AND INFORMATION
SYSTEMS DIVISION
PAPIERMUEHLESTRASSE 25
3003 BERNE
SWITZERLAND

Copies

ATTN DR R G LACAU 1
AEROSPATIALE MISSILE
DEPT E/ECN
CENTRE DES GATINES
91370 VERRIERE LE BUISSON
FRANCE

ATTN J M CHARBONNIER 1
VON KARMAN INSTITUTE
72 CHAUSSEE DE WATERLOO
1640 RHODE SAINT GENESE
BELGIUM

ATTN P CHAMPIGNY 1
DIRECTION DE L AERONAUTIQUE
ONERA
29 AV DE LA DIVISION LECLERC
92320 CHATILLON SOUS BAGNEUX CEDEX
FRANCE

ATTN DR P HENNIG 1
DEUTSCHE AEROSPACE (DASA)
VAS 414
ABWEHR AND SCHUTZ
POSTFACH 801149
8000 MUENCHEN 80
GERMANY

ATTN H G KNOCHE 1
DR GREGORIOU 1
MESSERSCHMIDT BOLKOW BLOHM
GMBH
UNTERNEHMENSBEREICH APPARATAE
MUNCHEN 80 POSTFACH 801149 BAYERN
GERMANY

ATTN DR S J YOON 1
AGENCY FOR DEFENSE DEVELOPMENT
AERODYNAMICS DIVISION (4-3-1)
P O BOX 35-4 YUSEONG TAEJON
KOREA

DISTRIBUTION (Continued)

Copies

ATTN PETER CAAP 1
HD FLIGHT SYS DEPT
FAA AERONAUTICAL RESEARCH INST
OF SWEDEN
BOX 11021
BROMMA SWEDEN 16111

ATTN DAVE BROWN 1
WEAPON SYSTEMS DIVISION
AERONAUTICAL AND MARITIME
RESEARCH LABORATORY
P O BOX 1500 SALISBURY
SOUTH AUSTRALIA 5108

INTERNAL

		Copies
B		1
B04		1
B04	(ZIEN)	1
B05	(GRAFF)	1
B05	(STATON)	1
B10		1
B10	(HSIEH)	1
B51	(ARMISTEAD)	1
B60	(TECHNICAL LIBRARY)	3
B60A	(GRAY)	1
C		1
D		1
G		1
G02		1
G04		5
G20		1
G205		1
G205	(ELLIOTT)	1
G205	(LONG)	1
G23		1
G23	(BIBEL)	1
G23	(CHADWICK)	1
G23	(COOK)	1
G23	(HANGER)	1
G23	(HARDY)	1
G23	(HOCK)	1
G23	(HYMER)	1
G23	(MALYEVAC)	1
G23	(OHLMEYER)	1
G23	(ROWLES)	1
G23	(WEISEL)	1
G30		1
G305		1
G32	(DAY)	1
G33	(MELTON)	1
G33	(RINALDI)	1
G50		1
G50	(SOLOMON)	1
G60		1
G70		1
G72		1
G72	(ALEXOPOULOS)	1
G72	(CHEPREN)	1
G72	(JONES)	1
G72	(ROBINSON)	5
G72	(MCINVILLE)	1
K		1
K40		1
K44	(ICHNIOWSKI)	1
K44	(JOHNSON)	1
N		1
N93	(NEGRON)	1
T		1
T406		1