



U.S. ARMY
RDECOM

TECHNICAL REPORT RDMR-SS-17-13

METHOD OF CHARACTERISTIC (MOC) NOZZLE FLOWFIELD SOLVER—USER'S GUIDE AND INPUT MANUAL VERSION 2.0

Kevin D. Kennedy

System Simulation and Development Directorate
Aviation and Missile Research, Development,
and Engineering Center

January 2018

Distribution Statement A: Approved for public release;
distribution is unlimited.



DESTRUCTION NOTICE

FOR CLASSIFIED DOCUMENTS, FOLLOW THE PROCEDURES IN DoD 5200.22-M, INDUSTRIAL SECURITY MANUAL, SECTION II-19 OR DoD 5200.1-R, INFORMATION SECURITY PROGRAM REGULATION, CHAPTER IX. FOR UNCLASSIFIED, LIMITED DOCUMENTS, DESTROY BY ANY METHOD THAT WILL PREVENT DISCLOSURE OF CONTENTS OR RECONSTRUCTION OF THE DOCUMENT.

DISCLAIMER

THE FINDINGS IN THIS REPORT ARE NOT TO BE CONSTRUED AS AN OFFICIAL DEPARTMENT OF THE ARMY POSITION UNLESS SO DESIGNATED BY OTHER AUTHORIZED DOCUMENTS.

TRADE NAMES

USE OF TRADE NAMES OR MANUFACTURERS IN THIS REPORT DOES NOT CONSTITUTE AN OFFICIAL ENDORSEMENT OR APPROVAL OF THE USE OF SUCH COMMERCIAL HARDWARE OR SOFTWARE.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing this collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503			
1. AGENCY USE ONLY	2. REPORT DATE January 2018	3. REPORT TYPE AND DATES COVERED Final	
4. TITLE AND SUBTITLE Method of Characteristic (MOC) Nozzle Flowfield Solver—User's Guide and Input Manual Version 2.0			5. FUNDING NUMBERS
6. AUTHOR(S) Kevin D. Kennedy			
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Commander, U.S. Army Research, Development, and Engineering Command ATTN: RDMR-SSM-A Redstone Arsenal, AL 35898-5000			8. PERFORMING ORGANIZATION REPORT NUMBER TR-RDMR-SS-17-13
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 A
13. ABSTRACT (Maximum 200 Words) The Method of Characteristic (MOC) Nozzle Flowfield Solver code is a structured, Two-Dimensional (2-D) (both planar and axisymmetric), isentropic supersonic flow solver for converging-diverging nozzle problems under choked flow conditions. The code uses a marching step method based on a second-order modified Euler predictor and corrector scheme with iteration. The initial starting line is determined either by a user-input profile or by Sauer's method for determining the sonic line. Initial wall options include either a user-input profile or a starting circular arc that can be followed by a second-order quadratic wall based on user inputs. The remaining downstream wall is then determined by the local characteristic and mass flow rate. A simple boundary-layer growth model is used to determine the viscous corrected shape. The software coding is written in FORTRAN 90/95. Input files are used to transfer information to the program.			
14. SUBJECT TERMS Method of Characteristic (MOC), Nozzle, Flowfield Solver			15. NUMBER OF PAGES 53
			16. PRICE CODE
17. SECURITY CLASSIFICATION OF REPORT UNCLASSIFIED	18. SECURITY CLASSIFICATION OF THIS PAGE UNCLASSIFIED	19. SECURITY CLASSIFICATION OF ABSTRACT UNCLASSIFIED	20. LIMITATION OF ABSTRACT SAR

TABLE OF CONTENTS

	<u>Page</u>
I. INTRODUCTION	1
II. PROGRAM READS AND WRITES.....	2
A. File Description.....	2
B. Program Reads	4
C. Program Writes	4
III. NAMELIST INPUTS	7
A. Overview.....	7
B. &INPUT Namelist Definition	8
IV. BOUNDARY LAYER MODEL.....	8
REFERENCES	11
LIST OF ACRONYMS AND ABBREVIATIONS.....	12
APPENDIX A: FLOW CHART.....	A-1
APPENDIX B: EXAMPLE 1.....	B-1
APPENDIX C: EXAMPLE 2	C-1
APPENDIX D: EXAMPLE 3	D-1
APPENDIX E: EXAMPLE 4.....	E-1
APPENDIX F: EXAMPLE 5.....	F-1
APPENDIX G: EXAMPLE 6	G-1
APPENDIX H: EXAMPLE 7	H-1
APPENDIX I: ERROR MESSAGES	I-1
APPENDIX J: NAMELIST INPUT SUMMARY	J-1

ILLUSTRATION

<u>Figure</u>	<u>Title</u>	<u>Page</u>
1.	BL Growth Model.....	9

LIST OF TABLES

<u>Table</u>	<u>Title</u>	<u>Page</u>
1.	Summary of Input and Output Files	3
2.	Namelist Input Summary.....	8

I. INTRODUCTION

The Method of Characteristic (MOC) Nozzle Flowfield Solver code is a FORTRAN 90/95 [1] structured, Two-Dimensional (2-D) (both planar and axisymmetric), isentropic supersonic flow solver for converging-diverging nozzle problems under choked flow conditions. It is based on the method laid out in References 2 and 3. The code uses a marching step method based on a second-order modified Euler predictor and corrector scheme with iteration. The initial starting line is determined by a user-input profile or by a method for determining the sonic line [4]. Initial wall options include either a user-input profile or a starting circular arc that can be followed by a second-order quadratic wall based on user inputs. The remaining downstream wall shape is then determined by the local characteristic and mass flow rate. A one-equation boundary-layer growth model is used to determine the viscous corrected shape. Input files are used to transfer information to the program.

In the early 1950s, the use of the MOCs was ideally suited to run on the early computer systems that were limited in speed and memory. As time went by and computer systems began to grow in performance, there was an ability to incorporate the full set of viscous equations of motion. These early Reynolds Averaged Navier-Stokes [5] programs quickly surpassed the Euler [6] limits of the MOCs. While these new codes were able to predict the full nozzle solution, they were not able to predict a nozzle shape that would result in a uniform nozzle exit flowfield. The objective of the MOC code is to predict nozzle contours for a given exit Mach number with uniform flow.

The MOC code was written as an analysis tool in science and engineering. This program was intended for users who needed to predict the nozzle flowfield and nozzle contour with uniform exit conditions. The code was not meant for the point-and-click community of computer users. The MOC code requires a command line interaction. No graphical user interface is provided with the computer program. The user is expected to have a basic knowledge of fluid dynamics in order to operate the code.

The MOC code was written for the Linux and UNIX [7, 8] Operating Systems (OS) required a FORTRAN 90/95 compiler and terminal window system, such as the X Window System [9]. The code was known to compile successfully with the Intel ifort [10] compiler, Version 15, and work well on Apple OS X [11] computer OS. The MOC memory and disk storage requirements are flexible and readily adjustable. Appendix A shows a flow chart with the top-level processes of the MOC code. It is advised that the user first execute the included test cases for comparison with the MOC code output, as shown in Appendixes B through H. This code is not guaranteed to run successfully or operate without failure on any given computer and/or OS. Considerable error messaging is included in the output of the code to aid the user in the understanding of its failure, as shown in Appendix I.

The MOC code was written to provide a user-friendly environment, while offering extensive analytical capabilities by means of the following features:

- All operational parameters are input using Namelist format. Additionally, the startline profile, if needed, is input through a separate file, as defined in Section II.
- Computational results from the program are output to the terminal window as the MOC code executes. This output is also sent to a formatted disk file known as the MOC.out.
- Plot files are generated in the Gnuplot format [12], Version 4.6, with the extension of *.dem. These files are set to produce Portable Document Format (PDF) output files when executed through the Gnuplot program.
- All of the FORTRAN 90/95 array sizes are set in a PARAMETER statement within the MOC code, making it relatively easy to find and change those array size limits, if required.
- The MOC code maintains an internal 64-bit precision (compiler options not needed); however, the code can be downgraded to single precision if needed.

II. PROGRAM READS AND WRITES

Section II describes the files that are used and generated by Version 2.0 of the MOC code. A description of the contents of each file is given in Section II.A.

A. File Description

The input and output files utilized by the code are listed in Table 1. The user must provide the files that are read by the MOC code. At the minimum, the user must supply an American Standard Code for Information Interchange (ASCII) NOZZLE.inp control file. An additional ASCII file that contains the nozzle starting transonic flow profile may be needed, depending on control options. The output ASCII files include the solution, data files containing flow properties at locations of interest, and associated Gnuplot files (a portable command-line driven graphing utility, Version 4.6 compatibility). Additionally, several unassigned FORTRAN 90/95 files (Units 7 through 11) may contain 2-D solutions at different locations within the nozzle.

Table 1. Summary of Input and Output Files

Filename	File Descriptor	Format	Program Read and Writes
NOZZLE.inp	Main input variables	ASCII	Read
Profile.dat	User input start line	ASCII	Read
CENTERLINE.dat	Centerline properties	ASCII	Write
CENTERLINE.dem	Centerline plot file	GNU	Write
SLOPE.dat	Wall slope	ASCII	Write
SLOPE_SMOOTHED.dat	Fitted wall slope	ASCII	Write
Startline.dat	Initial flow start line	ASCII	Write
MOC.out	Output of results	ASCII	Write
WALL.dat	Wall conditions	ASCII	Write
WALL.dem	Wall plot file	GNU	Write
WALL_BOUNDARY_CORRECTED.dat	BL wall	ASCII	Write
WALL_BOUNDARY_CORRECTED_SMOOTHED.dat	Corrected BL wall	ASCII	Write
WALL_SMOOTHED.dat	Fitted wall	ASCII	Write
STARTING_CONDITIONS.dem	Start line plot file	GNU	Write
WALL_DEFINITION.dem	Wall shape plot file	GNU	Write
X_Y_FUN.dem	Nozzle flow plot file	GNU	Write
fort.7	Start line solution	ASCII	Write
fort.8	Circular arc solution	ASCII	Write
fort.9	Second-order wall solution	ASCII	Write
fort.10	Final C+ Characteristic line	ASCII	Write
fort.11	Turning region flow	ASCII	Write

B. Program Reads

Section II.B details the files that are read into the MOC code. The user is required to generate these files.

1. NOZZLE.inp File

The NOZZLE.inp file is the main input file. It is in an ASCII FORTRAN 90/95 Namelist format. This file is required for all run options. It is composed of tolerances, control options, gas flow properties, nozzle wall definitions, and Boundary Layer (BL) controls. Details of this format are given in Section III.

2. Profile.dat File

The Profile.dat is dependent on user input. It contains the start line properties for the initial valued solution. It is a formatted ASCII file. This file must start with conditions on the centerline and move up in the direction of the wall. It must be in the format of I, J, K, X, Y, Z, Rho, U, V, W, and P where I, J, and K are integer index values (not used); X, Y, and Z (Z not used) are the axial, radial, and circumferential locations of each point; Rho is the density (kilograms per cubic meter (kg/m^3)); U, V, and W are the three velocity (meters per second (m/s)) components (W not used); and P is the pressure (newtons per square meters (N/m^2)). The temperature is calculated by the equation of state from the density and pressure. Each column must be free space delimited.

C. Program Writes

Section II.A details the files that are written out by the MOC code. The main source of output and errors are directed to Fortran Unit 6, which typically is the main terminal window from which the code was executed. This information is also captured in the file MOC.out.

1. CENTERLINE.dat File

The CENTERLINE.dat file is written out by the MOC code and contains the centerline properties. It is in the format of I, J, X, Y, U, V, Mach, Velocity Magnitude (Vel Mag), Theta, P, Rho, and T, where I is the marching integer index; J is the downward characteristic integer index; X and Y are the real axial and radial locations (meters (m)), respectively; U and V are the real axial and radial velocity (m/s), Mach is the real local Mach number; Vel Mag (m/s) is the real total velocity magnitude; Theta is the real local flow angle (degrees); P is the real pressure (N/m^2); Rho is the real density (kg/m^3); and T is the real temperature (Kelvin (K)). The radial location Y, V velocity, and flow angle in this file are defined to be of zero value. The first line in this file contains header information. The last axial location within this file is terminated when the centerline designed Mach number is reached, which may not be at the physical location of the end of the nozzle wall.

2. SLOPE.dat File

The SLOPE.dat file is a two-column formatted ASCII file containing the inviscid wall slope information. It is in the form of axial location (m) versus the wall slope.

3. SLOPE_SMOOTHED.dat File

The SLOPE_SMOOTHED.dat file is a two-column formatted ASCII file containing the inviscid wall slope information after it has been processed through the Least-Squared Curve Fit (LSCF) approximation. It is in the form of axial location (m) versus the wall slope. The order of this LSCF is a user-controlled input variable, IORDER, in namelist INPUT. The user needs to examine this file since the smoothed slope profile may not result in an expected profile (that is, a negative value when approaching zero).

4. WALL.dat File

The WALL.dat file is a multi-column formatted ASCII file containing the inviscid wall properties. It follows the same format as the CENTERLINE.dat file, as defined in Section II.C.1. The X, Y position (m) defines the inviscid wall location, and the flow angle (degrees) is the local wall angle. This file does not contain the correction needed to accommodate the BL thickness.

5. WALL_SMOOTHED.dat File

The WALL_SMOOTHED.dat file is a two-column formatted ASCII file containing the inviscid wall location after it has been processed through the LSCF approximation. The order of this LSCF is a user-controlled input variable, IORDER, in namelist INPUT. This is the same variable that is used to fit the SLOPE.dat file. This file is often used to represent the shape of the wall. It is smooth compared to the stair-stepping profile often observed in the WALL.dat file. This file does not contain the correction that is needed to accommodate the BL thickness.

6. WALL_BOUNDARY_CORRECTED.dat File

The WALL_BOUNDARY_CORRECTED.dat file is a multi-column, formatted ASCII file containing the BL-corrected file. This BL model is a simple one-equation model that adds a correction factor to the radial location of the wall based on local flow conditions and arc length. It contains an I, J index pair; the original inviscid X, Y point locations (m); normalized temperature (273.11 K based); viscosity (newton-seconds per square meter (N-s/m²)); Reynolds number; a one-eighth power law BL thickness (m) (not used); the new viscous corrected radial location (m); and the equivalent arc length (m) of the wall. This file is generally not plotted since the profile follows the same stair-step effect of the original wall.

7. WALL_BOUNDARY_CORRECTED_SMOOTHED.dat File

The WALL_BOUNDARY_CORRECTED_SMOOTHED.dat file is a two-column, formatted ASCII file containing the BL-corrected wall and X and new Y location (m) after it has been processed through the LSCF approximation. The order of this LSCF is a user-controlled input variable, IORDER, in namelist INPUT. This file is suitable for use as the corrected wall profile that is needed to compare with viscous numerical fluid flow solvers.

8. Startline.dat File

The Startline.dat file is an output formatted ASCII file containing properties and formats similar to the CENTERLINE.dat file, as shown in Section II.C.1. It is often used to examine the initial start line properties generated internally by the MOC program. The last column may or may not contain the running total of the mass flow rate (kilograms per second (kg/s)) depending on input options.

9. STARTING_CONDITIONS.dem File

The STARTING_CONDITIONS.dem file is in the Gnuplot format. GNU is a portable command-line driven graphing utility for Linux, OS/2, Microsoft (MS) Windows, Mac OS, and Virtual Memory System (VMS) platforms. The source code is copyrighted but freely distributed. It currently supports batch and command line mode operations. Reference 12 provides details concerning the Gnuplot file.

This plot file, when used with the Gnuplot program, is set up to generate a viewable plot file in PDF format named STARTING_CONDITIONS.pdf. It will contain three pages of plots. The first page contains four plots of starting radial pressure, temperature, density, and Mach number. The starting profile for each property is represented in red. From each red line there are separate black lines that represent the properties along the downward running characteristic starting at each point. The second page contains the two plots of velocity, axial and radial. This plot is useful for observing any waves in the starting solution. The last page is a single plot of flow angle along each characteristic. The user can use this information to determine if the initial solution is valid.

10. WALL_DEFINITION.dem File

The WALL_DEFINITION.dem file is in the Gnuplot format and is set up to generate a viewable plot file in PDF format named WALL_DEFINITION.pdf. It will contain the final wall shape for the inviscid solution. Since the user has several options to input an initial wall shape, this plot will show the difference in the initial versus the final wall shape. The transition points between the circular arc and the second-ordered wall are highlighted for clarity.

11. X_Y_FUN.dem File

The X_Y_FUN.dem file is in the Gnuplot format and is the main file used for viewing the results of the MOC code. This plot file, when used with the Gnuplot program, is set up to generate up to eight different pages of plots. These plots are in PDF format. The file is named X_Y_FUN.pdf. The first plot displays the matching locations for each section of the nozzle. Each point is highlighted by a symbol. The second plot is identical to the first plot but without the symbol markers. This makes the plot easier to understand if the input resolution is set very high. The next plot file is a colored image of Mach number flow within the nozzle. If the resolution is coarse, this image may not appear correctly. This is largely a limit on cell shading within the Gnuplot program. Fine-resolution plots should not have this problem. Additionally, gaps may appear within the plot that represent flow transition that results from changing wall type (that is, circular to second-order). The fourth and fifth pages are of axial and radial velocity (m/s), respectively. The sixth page is of flow angle (degrees). Strong waves with

the flowfield are apparent in this plot. The seventh page is a line plot of the wall shape. The four curves are of the original and smooth profiles of the viscous and inviscid solutions. This plot is useful for examining the BL thickness and the effects of LSCF smoothing of the initial profile. The last plot is the slope of the original wall and the slope of the smoothed wall. This plot is useful for determining where the waves within the nozzle are hitting the wall, which cause a sudden change in slope.

12. CENTERLINE.dem File

The CENTERLINE.dem file is in the Gnuplot format and is set up to generate a two-page PDF file named CENTERLINE.pdf. The first page contains four line plots of centerline conditions of pressure, temperature, density, and Mach number. The second page is a single line plot of axial velocity.

13. WALL.dem File

The WALL.dem file is also in the Gnuplot format and is set up to generate a two-page PDF file named WALL.pdf. It is identical in format to the centerline plot (Section II.C.12) but displays conditions along the nozzle wall. The first page contains four line plots of wall conditions of pressure, temperature, density, and Mach number. The second page is a single line plot of axial velocity.

14. fort.7, fort.8, fort.9, fort.10, and fort.11 Files

The fort.7, fort.8, fort.9, fort.10, and fort.11 files contain the intermediate data needed to generate the PDF plot files and are not intended to be used by the user. Once the users generate the PDF files, fort files can be safely removed.

III. NAMELIST INPUTS

A. Overview

The main MOC input file, NOZZLE.inp, contains user-specified values arranged in a FORTRAN 90/95 format. A namelist is a method of ordering input data into a routine. The name of the namelist is preceded by the &INPUT character string, which starts in the second column of the input file. The namelist is terminated by either a Forward Slash (/) symbol or &END, and also at the beginning in the second column of the input file. Comments may be placed within the file at any position but must be preceded with the Exclamation (!) symbol.

There is only one namelist group. This group will define the options for different features, including code operation, flowfield and boundary conditions, and output. All supported variables in the namelist will be described, along with any default information.

B. &INPUT Namelist Definition

Namelist &INPUT is used to define runtime parameters and reference quantities. It is read only once from the NOZZLE.inp file and contains several parameters. To simplify the definition of parameters, they have been grouped and are listed in Table 2. Appendix J provides more detailed information concerning tolerance controls, control values, gas properties, nozzle wall properties, and nozzle solution parameters.

Table 2. Namelist Input Summary

Tolerance	Control Values	Gas Properties	Nozzle Properties	Nozzle Solution
ICOR	DELTA	PS	YT	EMD
E1	NI	TS	RTU	BL SCALE
E2	NT	PA	RTD	I_FINAL
E3	KWRITE	G	AA	NOZ
E4	IUNITS	RG	AE	IVL
E5	ST	GL	XE	
	SMOOTH	GC		
	IORDER	VAR_GAMMA		

IV. BOUNDARY LAYER MODEL

Simple flat plate BLs with little to no pressure gradient can be based on the local Reynolds number (the ratio of inertial to viscous forces). Flat plate BL models are typically some function of the Reynolds number raised to a power (power law). In a nozzle, the pressure gradient can be severe. This makes a Reynolds number-based BL difficult to model. Nozzle shapes with a sudden large expansion near the throat will have a large drop in pressure that will suppress the growth of the BL. Likewise, nozzle shapes with a sudden turn down in the wall will have an enhanced BL thickness.

It would be possible to code a multi-equation model that tracks the rate of growth and thickness of the BL within the code. Due to complexity and time, this was not done for the MOC code. The BL thickness prediction in the MOC is a simple one-equation growth model based on the arc length of the wall. This model is called within a subroutine that could be easily replaced in the future. This model simply adds a thickness to the MOC predicted wall as a function of arc length. The model is a best-fit, tenth-order LSCF based on the thickness of the BLs in each of the test cases when compared to the CRAFT Computational Fluid Dynamics (CFD) [13] code. The CRAFT code is a structured grid Navier-Stokes solver with an advanced two-equation k- ϵ viscous solver.

To better understand this model, the figure displays on a log and log scale the length and thickness of the BL from each of the various test cases. These nozzle predictions range in length from a few meters to hundreds of meters in length. The black line is the MOC model thickness as a function of length. Each of the thicknesses from the test cases is overlaid on the plot. The stair-step appearance of thickness from each model is due to the size and shape of the numerical grid used in each CRAFT CFD modeled case. Note that Test Cases 1 and 2 had a user-input scale factor greater than one, as shown in Appendixes B and C, which resulted in each thickness being greater than the predicted MOC model. This was done to show that the user can enhance the thickness of the BL, if needed. As previously mentioned, if the nozzle wall suddenly expands or if the nozzle shape produces shock waves that interact with the BL (sudden increase in pressure on the wall), the MOC model may need to be scaled.

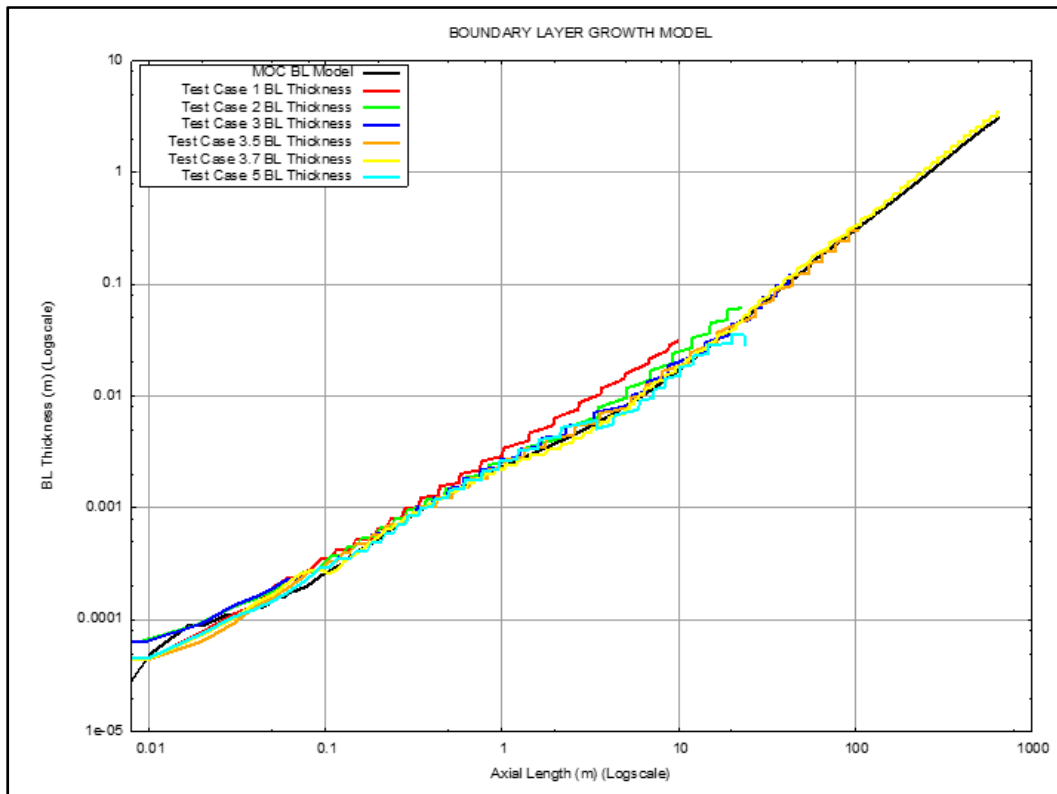


Figure. BL Growth Model

REFERENCES

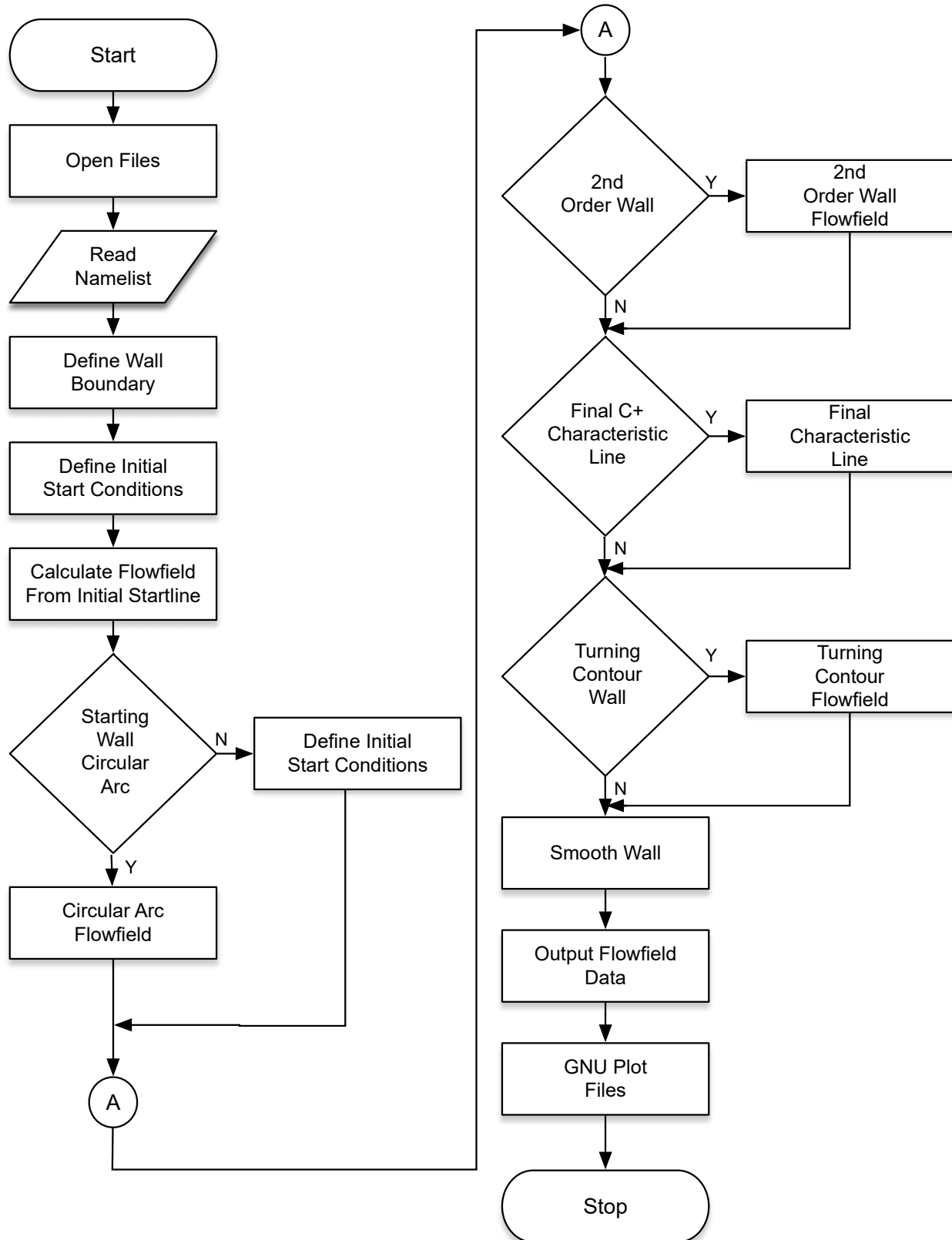
1. Adams, J. C. et al., Fortran 95 Handbook, The Massachusetts Institute of Technology (MIT) Press, Cambridge, MA, 1997.
2. Zucrow, M. J. and Hoffman, J. D., Gas Dynamics, Volume I, John Wiley & Sons, Inc., New York, 1976.
3. Zucrow, M. J. and Hoffman, J. D., Gas Dynamics, Volume II, John Wiley & Sons, Inc., New York, 1977.
4. Sauer, R., “General Characteristics of the Flow Through Nozzles at Near Critical Speeds,” NACA-TM-1147, National Advisory Committee for Aeronautics (NACA), Washington, DC, 30 June 1947.
5. “Reynolds-averaged Navier-Stokes equations,” Wikipedia, September 2017, https://en.wikipedia.org/wiki/Reynolds-averaged_Navier%E2%80%93Stokes_equations.
6. “Euler Equations,” Wikipedia, September 2017, [https://en.wikipedia.org/wiki/Euler_equations_\(fluid_dynamics\)](https://en.wikipedia.org/wiki/Euler_equations_(fluid_dynamics)).
7. Bovet, D. and Cesati, M., Understanding the LINUX Kernel, 3rd Edition, O'Reilly Media, Inc., 2005.
8. Rosen, K. et al., UNIX: The Complete Reference, 2nd Edition, McGraw-Hill, 2007.
9. Young, D.A., The X Window System: Programming and Applications with Xt, OSF/Motif, 2nd Edition, Prentice-Hall, 1994.
10. “Intel Fortran Compiler,” September 2017, <http://software.intel.com>.
11. “Mac OS X,” Wikipedia, September 2017, <https://en.wikipedia.org/wiki/MacOS>.
12. GNU PLOT Homepage, January 2012, <http://www.gnuplot.info>.
13. “Products – CRAFT CFD,” CRAFT-TECH Homepage, 2017, <https://www.craft-tech.com/products/craft-cfd/>.

LIST OF ABBREVIATIONS, ACRONYMS, AND SYMBOLS

/	Forward Slash
!	Exclamation
2-D	Two Dimensional
ASCII	American Standard Code for Information Interchange
BL	Boundary Layer
CFD	Computational Fluid Dynamics
Chara	Characteristic
K	Kelvin
kg/m ³	kilograms per cubic meter
kg/s	kilogram per second
LSCF	Least-Squared Curve Fit
m/s	meters per second
m	meter
MOC	Method of Characteristic
MS	Microsoft
N/m ²	newtons per square meter
N-s/m ²	newtons-second per square meter
OS	Operating System
PDF	Portable Document Format
VEL MAG	Velocity Magnitude
VMS	Virtual Memory System

**APPENDIX A
FLOW CHART**

The flow chart is offered to show the top-level processes of the Method of Characteristic (MOC) code.



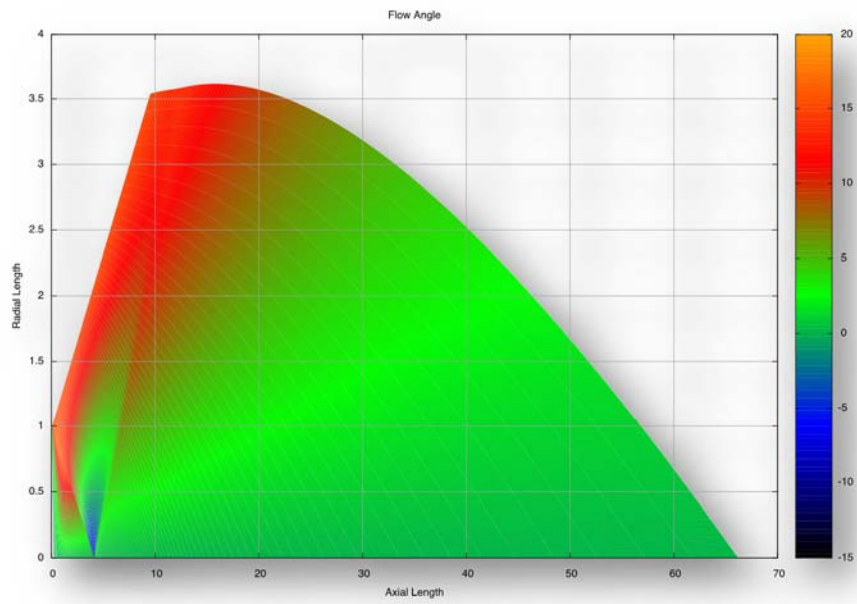
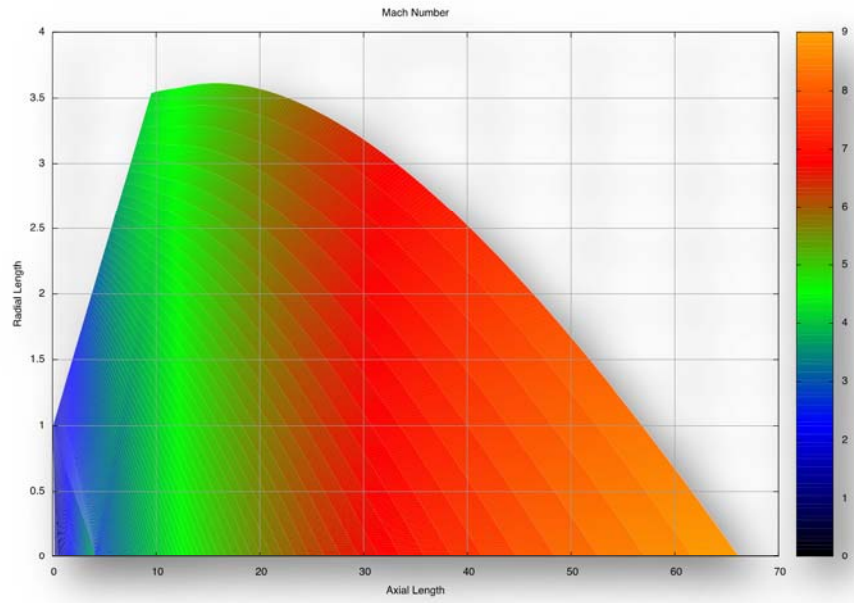
APPENDIX B
EXAMPLE 1

Example 1 is set up to model an axisymmetric nozzle. The code will calculate a transonic start line based on input conditions. The nozzle wall is defined at all locations with a circular arc and a second-order curve. The flowfield calculation along the wall is stopped at 10 m before the centerline design Mach number of 5.0 is reached, however the centerline calculation continues on to 65 m. Data smoothing is turned on. The boundary layer thickness is increased by a scale factor of 1.77. This case is designed to emphasize the presence of the development of strong waves near the centerline. These waves result in grid lines crossing at about 5-meters downstream. The user should take note of the variables highlighted in red. Plots of the Mach number and flow angle are included.

```

&INPUT
!...
!... Set Tolerances
!...
      ICOR = 120      !predictor corrector term
      E1  = 1.0E-10 !m X space error
      E2  = 1.0E-10 ! Pressure error
      E3  = 1.0E-10 ! Density error
      E4  = 1.0E-10 ! Velocity error
      E5  = 1.0E-10 ! flow angle error
!...
!... Set Control values
!...
      DELTA = 1.0      !1 axi, 0 planer (Mass flux not working correctly)
      NI    = 81      !NUMBER OF RADIAL POINTS ON INFLOW PLANE (Max 99)
      NT    = 31      !NUMBER OF CIRULAR ARC POINTS
      KWRITE = 1
      IUNITS = 2
      SMOOTH = .TRUE. !EVENLY SPACE OUT THE DATA AFTER EACH SECTION
      ST    = 1.0      !0.0 Even space start line, 1.0 quadratic spaced
      IORDER = 15
!...
!... Set Gas Properties
!...
      PS    = 1600.000E+06 !gas pressure
      TS    = 1.000000E+03 !gas temperature
      PA    = 0.0          !ambient pressure
      G     = 1.4          !gamma
      RG    = 287.040     !J/KG-K
      GC    = 1.0          !M-KG/N-S^2
      GL    = 1.0          !M^2/M^2
!...
!... Set Nozzle Properties
!...
!... FOR A NOZZLE HAVING A CIRCULAR ARC THROAT JOINED TANGENTIALLY TO
!... A WALL HAVING A CONTOUR DESCRIBED BY A SECOND ORDER POLYNOMIAL -
!... LOCATE THE INTERSECTION OF A LEFT RUNNING CHARACTERISTIC
!... WITH A SECOND ORDER QUADRATIC WALL CONTOUR
!...
!... INPUT:
!...      YT is the throat radius
!...      RTD is the radius of curvature of throat above YT
!...      AA is angle for RTD
!...      AE is angle at lip
!...      XE is the exit location
!...
!...
!...      *
!...      * <- Angle AA      - <- Exit Location XE with angle AE
!...      *
!...      *      -
!...      *      -
!...      *      -
!...      *      -
!...      v      - Attachment Point
!... THROAT - - -
!...      ^

```

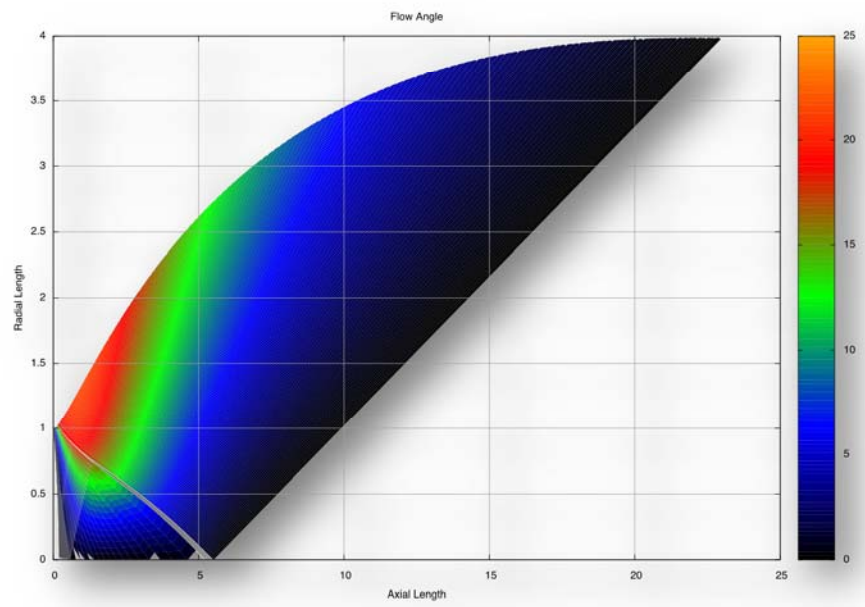
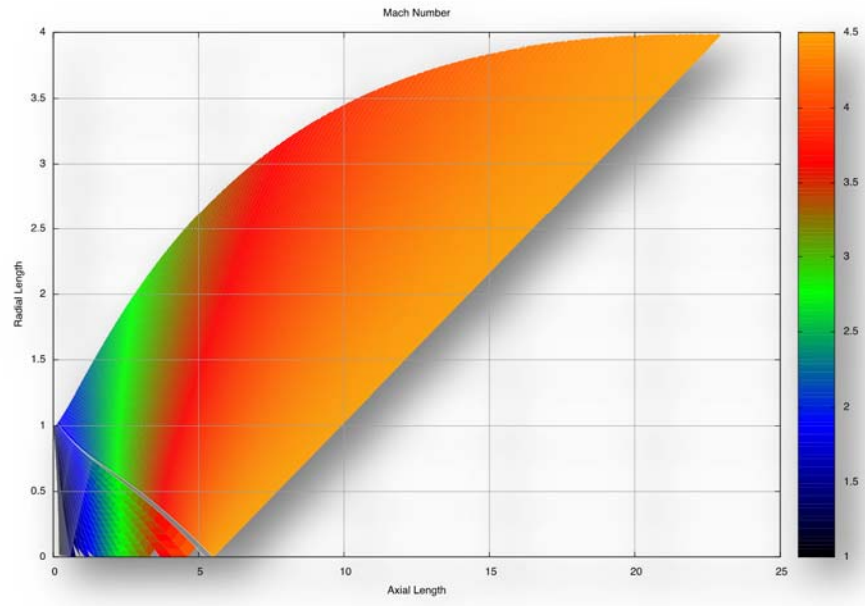
APPENDIX C
EXAMPLE 2

Example 2 is set up to model an axisymmetric nozzle. The code will calculate a transonic start line based on input conditions. The nozzle wall is initially defined with a circular arc. The flowfield calculation is stopped when the design Mach number of 4.5 on the centerline is reached. There is no data smoothing. This case is designed to emphasize the large attachment angle and the creation of the inviscid wall. The user should take note of the variables highlighted in red. Plots of the Mach number and flow angle are included.

```

&INPUT
!...
!... Set Tolerances
!...
    ICOR = 80      !predictor corrector term
    E1  = 1.0E-10 !m X space error
    E2  = 1.0E-10 ! Pressure error
    E3  = 1.0E-10 ! Density error
    E4  = 1.0E-10 ! Velocity error
    E5  = 1.0E-10 ! flow angle error
!...
!... Set Control values
!...
    DELTA = 1.0      !1 axi, 0 planer
    NI    = 71      !NUMBER OF RADIAL POINTS ON INFLOW PLANE (Max 99)
    NT    = 35      !NUMBER OF CIRULAR ARC POINTS
    KWRITE = 1
    IUNITS = 2
    SMOOTH = .FALSE. !EVENLY SPACE OUT THE DATA AFTER EACH SECTION
    ST    = 1.0      !0.0 Even space start line, 1.0 quadratic spaced
    IORDER = 30
!...
!... Set Gas Properties
!...
    PS    = 1600.000E+06 !gas pressure
    TS    = 1.000000E+03 !gas temperature
    PA    = 0.0          !ambient pressure
    G     = 1.4          !gamma
    RG    = 287.040      !J/KG-K
    GC    = 1.0          !M-KG/N-S^2
    GL    = 1.0          !M^2/M^2
!...
!... Set Nozzle Properties
!...
    YT    = 1.000 !nozzle radius
    RTU   = 2.000 !radius of curve upstream of throat
    RTD   = 0.500 !radius of curve downstream of throat
    AA    = 35.0    !attachment angle
    AE    = 15.0    !exit angle
    XE    = 25.0    !exit location
!...
!... Set Nozzle Solution
!...
    IVL   = 0      !0 transonic start, 1 User defined input
    EMD   = 4.5    !design mach number
    NOZ   = 0      !0 calculate a MOC wall, 1 no MOC wall calculated
    I_FINAL = 351  !number of points on final chara line
    BL_SCALE = 1.35 !boundary layer scalar
/

```

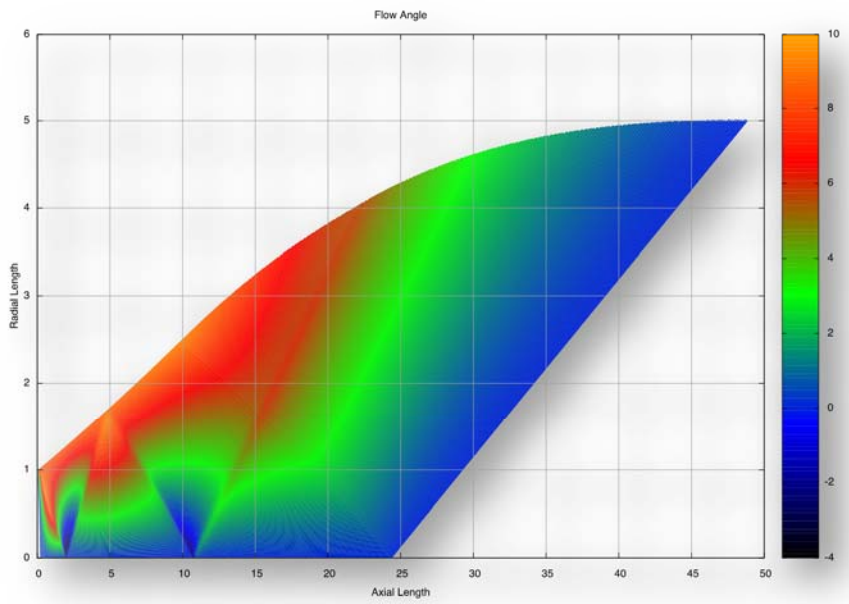
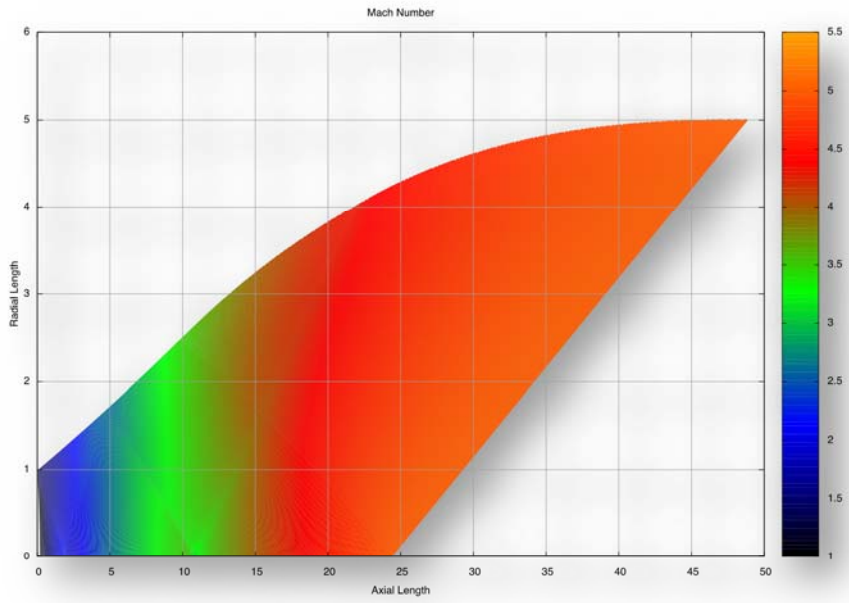
APPENDIX D
EXAMPLE 3

Example 3 is set up to model an axisymmetric nozzle. The code will calculate a transonic start line based on input conditions. The nozzle wall is initially defined with a circular arc, then a second-order quadratic, and then a calculated inviscid wall. The flowfield calculation is stopped when the design Mach number on the centerline is reached. Data smoothing is on. This case is designed to emphasize how the inviscid wall cancels out the strong wave in the flowfield. The user should take note of the variables highlighted in red. Plots of the Mach number and flow angle are included.

```

&INPUT
!...
!... Set Tolerances
!...
    ICOR = 80      !predictor corrector term
    E1  = 1.0E-10 !m X space error
    E2  = 1.0E-10 ! Pressure error
    E3  = 1.0E-10 ! Density error
    E4  = 1.0E-10 ! Velocity error
    E5  = 1.0E-10 ! flow angle error
!...
!... Set Control values
!...
    DELTA = 1.0      !1 axi, 0 planer
    NI    = 71      !NUMBER OF RADIAL POINTS ON INFLOW PLANE (Max 99)
    NT    = 31      !NUMBER OF CIRULAR ARC POINTS
    KWRITE = 1
    IUNITS = 2
    ST    = 1.0      !0.0 Even space start line, 1.0 quadratic spaced
    SMOOTH = .TRUE. !EVENLY SPACE OUT THE DATA AFTER EACH SECTION
!...
!... Set Gas Properties
!...
    PS    = 1600.000E+06 !gas pressure
    TS    = 1.000000E+03 !gas temperature
    PA    = 0.0        !ambient pressure
    G     = 1.4        !gamma
    RG    = 287.040    !J/KG-K
    GC    = 1.0        !M-KG/N-S^2
    GL    = 1.0        !M^2/M^2
!...
!... Set Nozzle Properties
!...
    YT    = 1.000 !nozzle radius
    RTU   = 2.000 !radius of curve upstream of throat
    RTD   = 1.000 !radius of curve downstream of throat
    AA    = 8.0     !attachment angle
    AE    = 15.0    !exit angle
    XE    = 55.0    !exit location
!...
!... Set Nozzle Solution
!...
    IVL   = 0      !0 transonic start, 1 User defined input
    EMD   = 5.0    !design mach number
    NOZ   = 0      !1 nozzle shape inputted, 0 nozzle shape from MOC
    I_FINAL = 301  !number of points on final chara line
    BL_SCALE = 1.0 !boundary layer scalar
    IORDER = 45
/

```



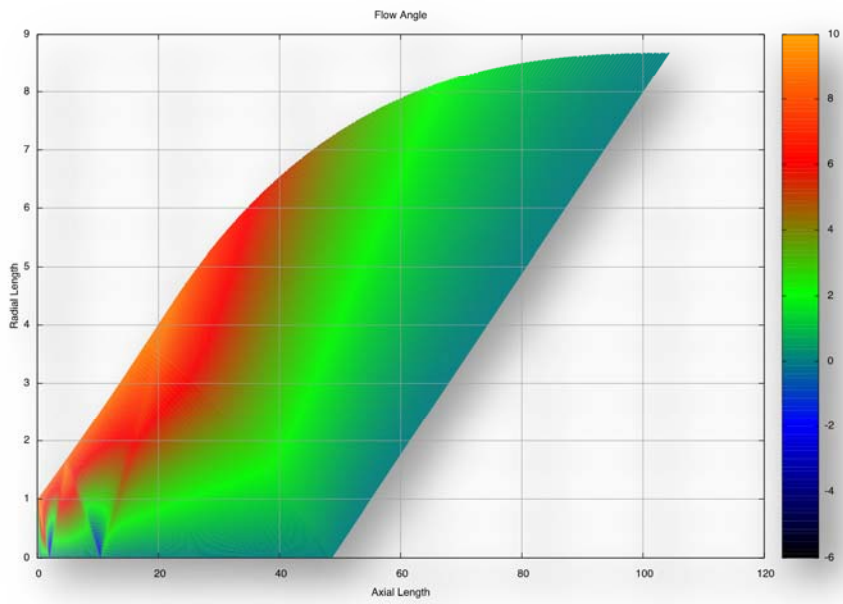
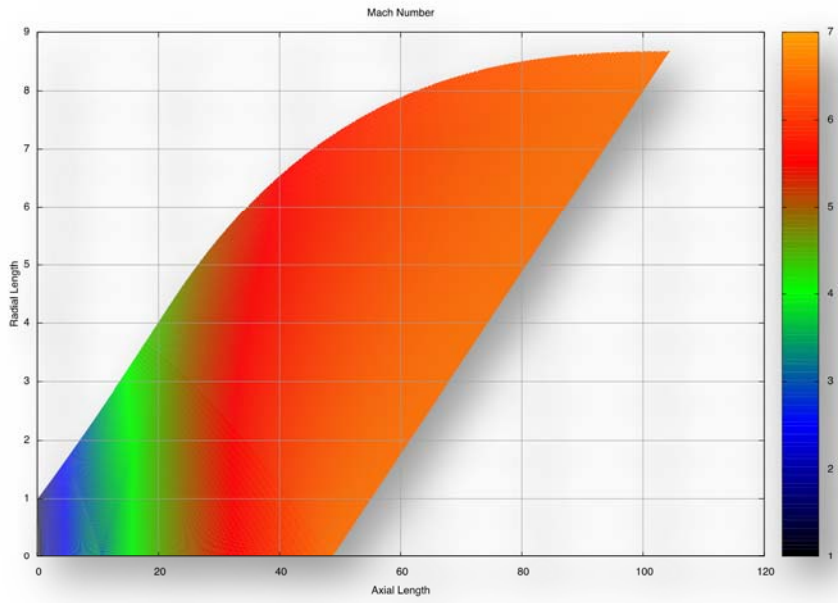
APPENDIX E
EXAMPLE 4

Example 4 is a modification of example 3. The main changes include a design Mach number increased to 6.5 from the original 5.0. The number of points on the initial startline and circular arc wall were changed. This case is designed to emphasize the effects of the longer nozzle due to the higher design Mach number. The user should take note of the variables highlighted in red. Plots of the Mach number and flow angle are included.

```

&INPUT
!...
!... Set Tolerances
!...
      ICOR = 80      !predictor corrector term
      E1  = 1.0E-10 !m X space error
      E2  = 1.0E-10 ! Pressure error
      E3  = 1.0E-10 ! Density error
      E4  = 1.0E-10 ! Velocity error
      E5  = 1.0E-10 ! flow angle error
!...
!... Set Control values
!...
      DELTA = 1.0      !1 axi, 0 planer
      NI    = 51      !NUMBER OF RADIAL POINTS ON INFLOW PLANE (Max 99)
      NT    = 41      !NUMBER OF CIRULAR ARC POINTS
      KWRITE = 1
      IUNITS = 2
      ST    = 1.0      !0.0 Even space start line, 1.0 quadratic spaced
      SMOOTH = .TRUE. !EVENLY SPACE OUT THE DATA AFTER EACH SECTION
!...
!... Set Gas Properties
!...
      PS    = 1600.000E+06 !gas pressure
      TS    = 1.000000E+03 !gas temperature
      PA    = 0.0          !ambient pressure
      G     = 1.4          !gamma
      RG    = 287.040     !J/KG-K
      GC    = 1.0          !M-KG/N-S^2
      GL    = 1.0          !M^2/M^2
!...
!... Set Nozzle Properties
!...
      YT    = 1.000 !nozzle radius
      RTU   = 2.000 !radius of curve upstream of throat
      RTD   = 1.000 !radius of curve downstream of throat
      AA    = 8.0      !attachment angle
      AE    = 15.0     !exit angle
      XE    = 155.0    !exit location
!...
!... Set Nozzle Solution
!...
      IVL   = 0          !0 transonic start, 1 User defined input
      EMD   = 6.5        !design mach number
      NOZ   = 0          !1 nozzle shape inputted, 0 nozzle shape from MOC
      I_FINAL = 301     !number of points on final chara line
      BL_SCALE = 1.0    !boundary layer scalar
      IORDER = 45
/

```



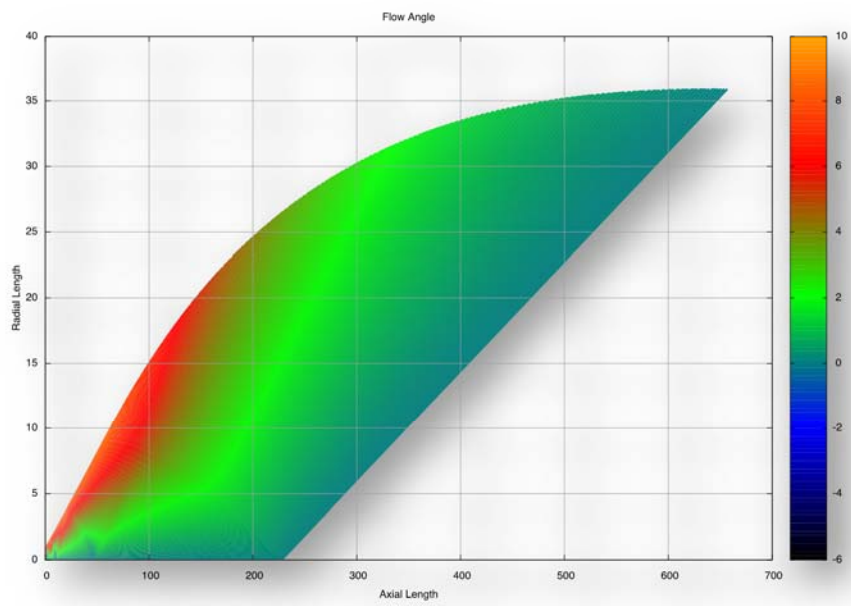
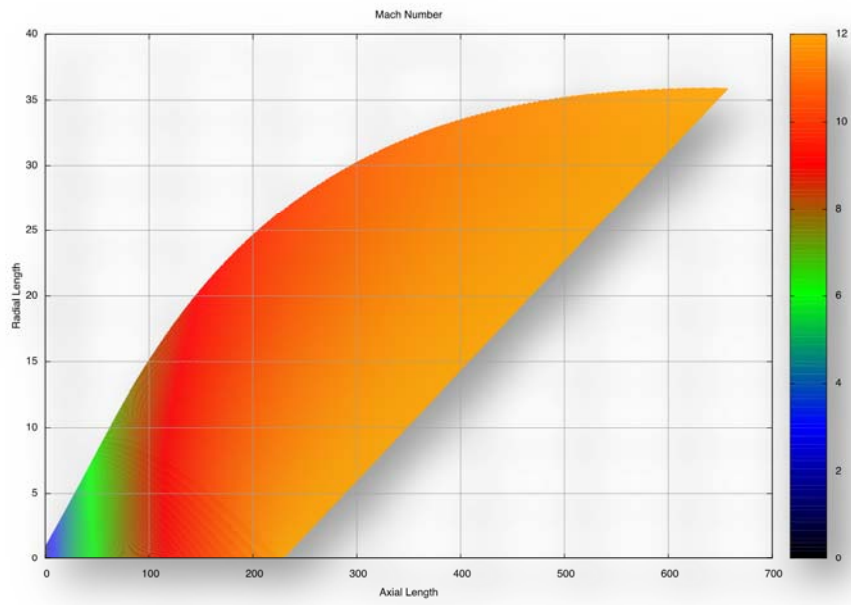
APPENDIX F
EXAMPLE 5

Example 5 is a modification of examples 3 and 4. The main changes include a design Mach number increased to 12.0. This case is designed to emphasize the effects of the longer nozzle due to the higher design Mach number. The user is encouraged to compare the results from examples 3, 4, and 5. Plots of the Mach number and flow angle are included.

```

&INPUT
!...
!... Set Tolerances
!...
    ICOR = 80      !predictor corrector term
    E1  = 1.0E-10 !m X space error
    E2  = 1.0E-10 ! Pressure error
    E3  = 1.0E-10 ! Density error
    E4  = 1.0E-10 ! Velocity error
    E5  = 1.0E-10 ! flow angle error
!...
!... Set Control values
!...
    DELTA = 1.0      !1 axi, 0 planer
    NI    = 51      !NUMBER OF RADIAL POINTS ON INFLOW PLANE (Max 99)
    NT    = 41      !NUMBER OF CIRULAR ARC POINTS
    KWRITE = 1
    IUNITS = 2
    ST    = 1.0      !0.0 Even space start line, 1.0 quadratic spaced
    SMOOTH = .TRUE. !EVENLY SPACE OUT THE DATA AFTER EACH SECTION
!...
!... Set Gas Properties
!...
    PS    = 1600.000E+06 !gas pressure
    TS    = 1.000000E+03 !gas temperature
    PA    = 0.0        !ambient pressure
    G     = 1.4        !gamma
    RG    = 287.040    !J/KG-K
    GC    = 1.0        !M-KG/N-S^2
    GL    = 1.0        !M^2/M^2
!...
!... Set Nozzle Properties
!...
    YT    = 1.000 !nozzle radius
    RTU   = 2.000 !radius of curve upstream of throat
    RTD   = 1.000 !radius of curve downstream of throat
    AA    = 8.0    !attachment angle
    AE    = 15.0  !exit angle
    XE    = 700.0 !exit location
!...
!... Set Nozzle Solution
!...
    IVL   = 0      !0 transonic start, 1 User defined input
    EMD   = 12.0   !design mach number
    NOZ   = 0      !1 nozzle shape inputted, 0 nozzle shape from MOC
    I_FINAL = 301  !number of points on final chara line
    BL_SCALE = 1.0 !boundary layer scalar
    IORDER = 15
/

```



APPENDIX G
EXAMPLE 6

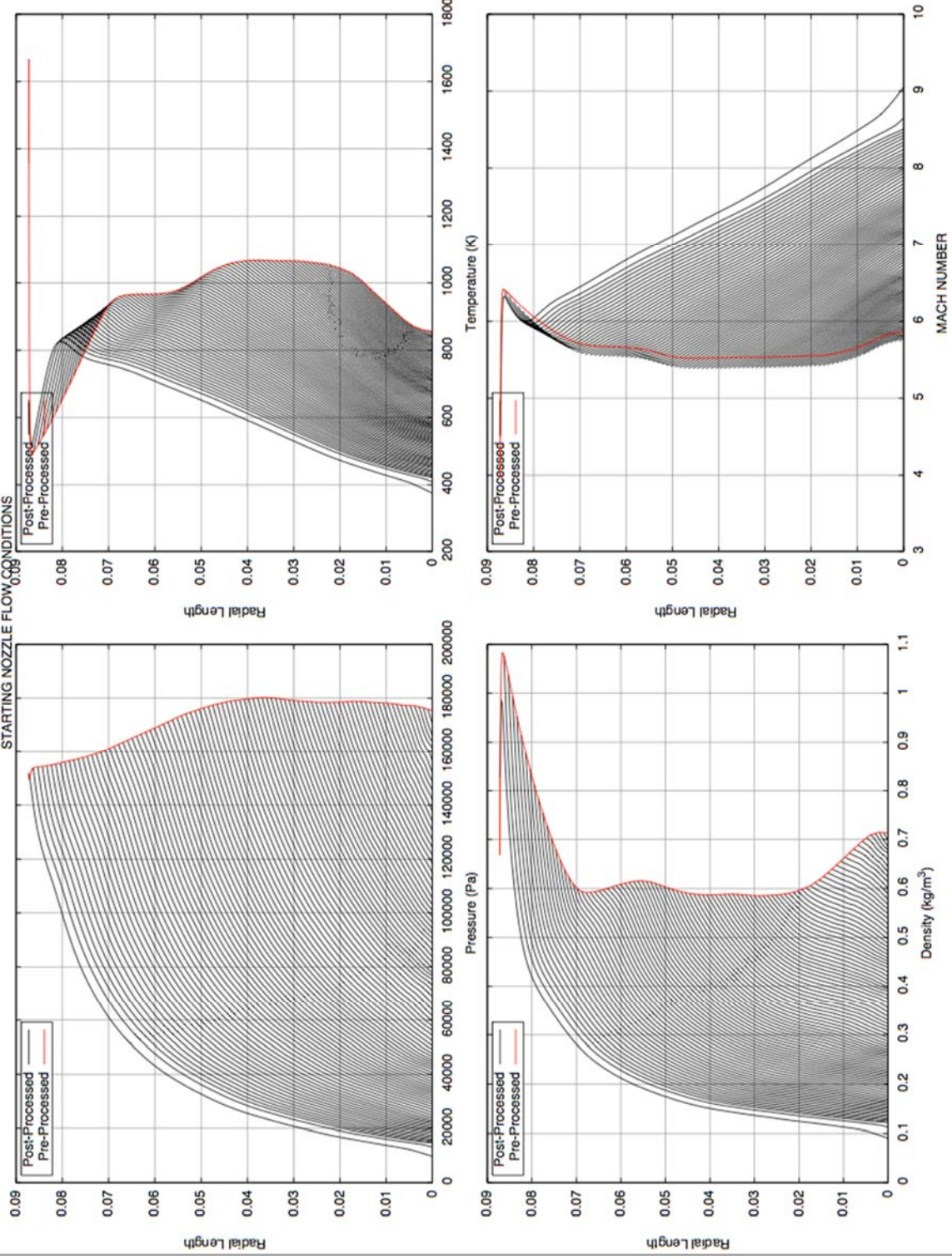
Example 6 is set up to model an axisymmetric nozzle. The user will specify the start line input conditions. The nozzle wall is initially defined with a circular arc, then a second-order quadratic, and then a calculated inviscid wall. The flowfield calculation is stopped when the design Mach number on the centerline is reached. Data smoothing is on. This case is designed to show that the **MOC** code can start with an arbitrary start line. The exit solution near the wall does not match the design Mach number. This is due to a cold profile that is near the wall. The user should take note of the variables highlighted in red. Plots of the start line (including its initial flowfield solution), Mach number, and flow angle are included.

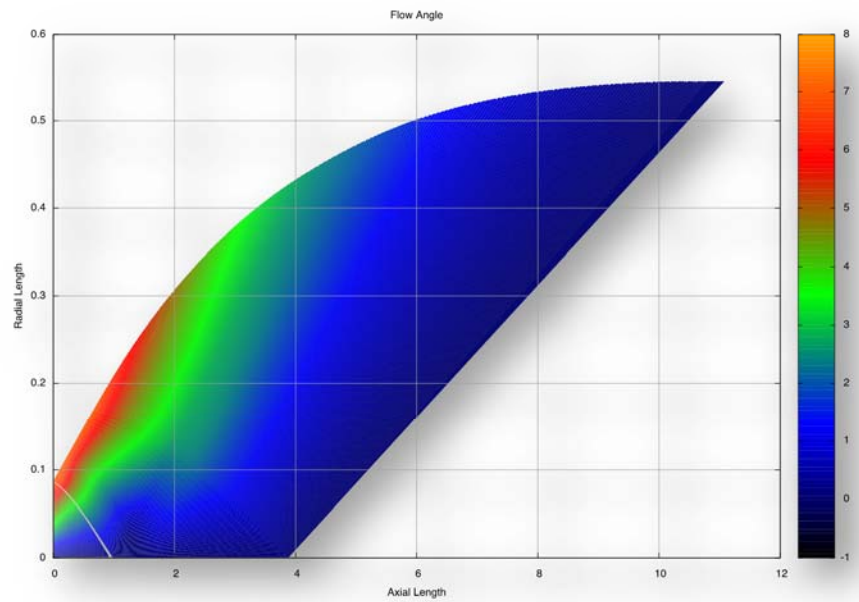
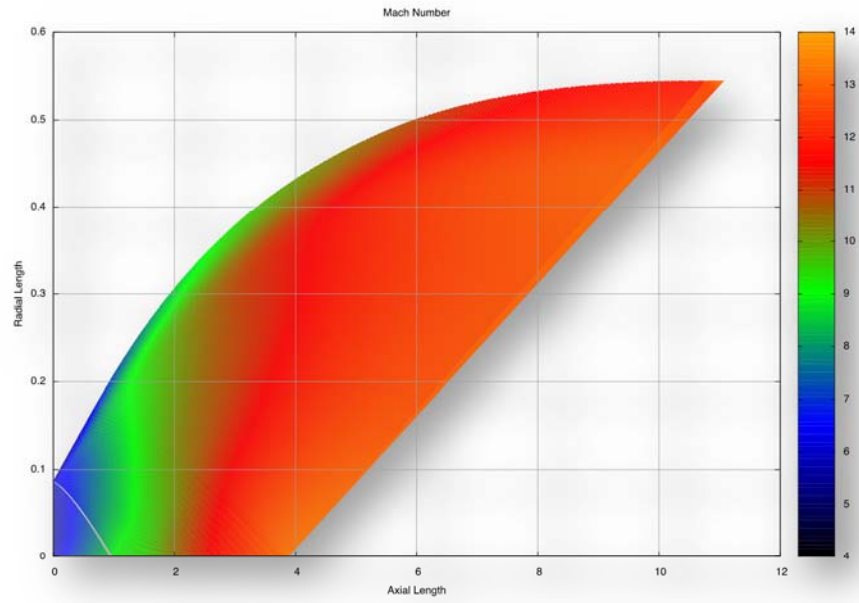
```

&INPUT
!...
!... Set Tolerances
!...
      ICOR = 80      !predictor corrector term
      E1  = 1.0E-10 !m X space error
      E2  = 1.0E-10 ! Pressure error
      E3  = 1.0E-10 ! Density error
      E4  = 1.0E-10 ! Velocity error
      E5  = 1.0E-10 ! flow angle error
!...
!... Set Control values
!...
      DELTA = 1.0      !1 axi, 0 planer
      NI    = 91      !NUMBER OF RADIAL POINTS ON INFLOW PLANE (Max 99)
      NT    = 15      !NUMBER OF CIRULAR ARC POINTS
      KWRITE = 1
      IUNITS = 2
      SMOOTH = .TRUE. !EVENLY SPACE OUT THE DATA AFTER EACH SECTION
      ST    = 1.0      !0.0 Even space start line, 1.0 quadratic spaced
      IORDER = 25
!...
!... Set Gas Properties
!...
      PS    = 1600.000E+06 !gas pressure chamber
      TS    = 1.000000E+03 !gas temperature chamber
      PA    = 0.0         !ambient pressure
      G     = 1.4         !gamma
      RG    = 287.040     !Gas constant J/KG-K
      GC    = 1.0         !M-KG/N-S^2
      GL    = 1.0         !M^2/M^2
!...
!... Set Nozzle Properties
!...
      YT    = 1.000 !nozzle radius
      RTU   = 2.000 !radius of curvature upstream of throat
      RTD   = 0.500 !radius of curvature downstream of throat
      AA    = 15.0    !attachment angle - angle at which flow transition to MOC wall
      AE    = 6.259  !exit angle
      XE    = 12.00  !exit location
!...
!... Set Nozzle Solution
!...
      IVL   = 1      !0 transonic start, 1 User defined input from "Profile.dat"
      EMD   = 13.25  !design exit mach number
      NOZ   = 0      !1 nozzle shape inputted, 0 nozzle shape from MOC
      I_FINAL = 351  !number of points on final chara line
      BL_SCALE = 1.0 !boundary layer scalar
/

```

STARTING NOZZLE FLOW CONDITIONS





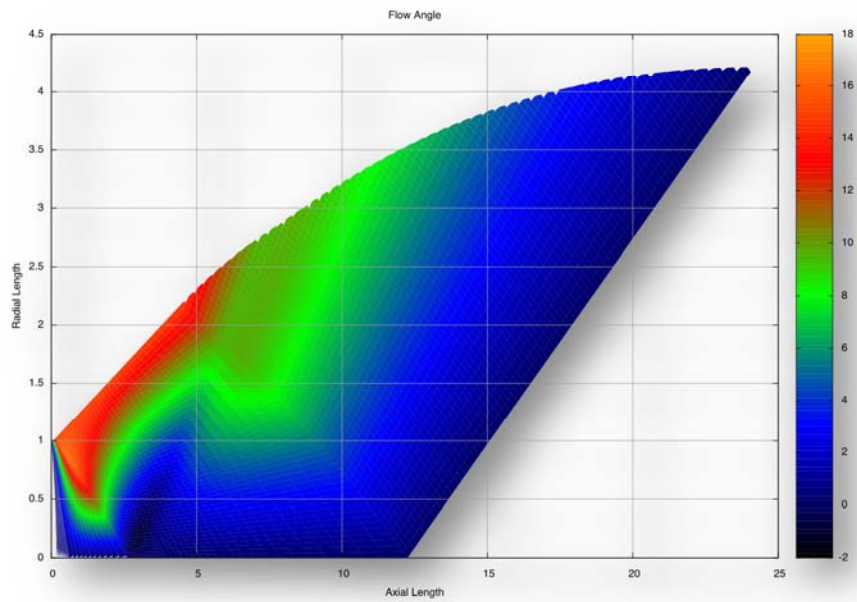
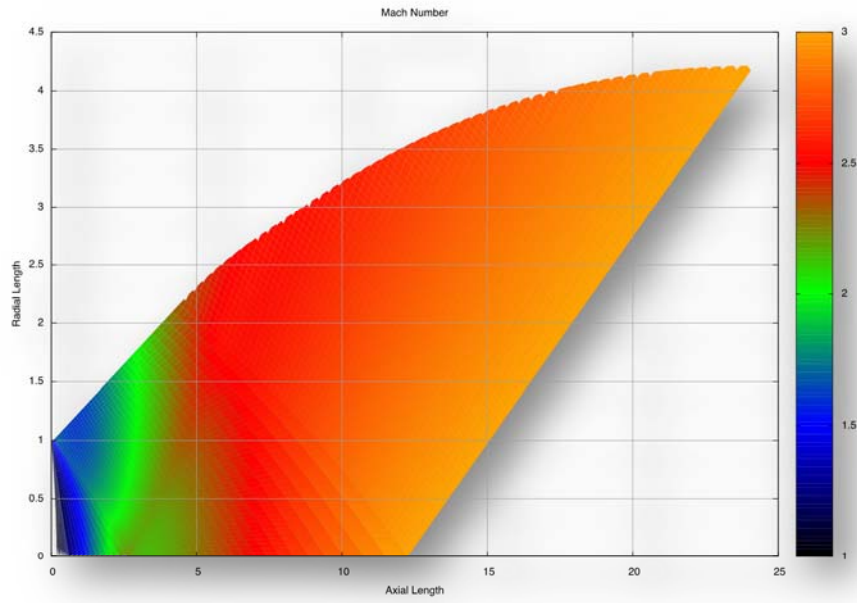
APPENDIX H
EXAMPLE 7

Example 7 is set up to model a planar nozzle. The code will calculate a transonic start line based on input conditions. The nozzle wall is initially defined with a circular arc, then a second-order quadratic, and then a calculated inviscid wall. The flowfield calculation is stopped when the design Mach number on the centerline is reached. Data smoothing is on. This case is designed to emphasize how the inviscid wall cancels out the strong wave in the flowfield and to show that planar nozzles have large exit radii. The user should take note of the variables highlighted in red. Plots of the Mach number and flow angle are included.

```

&INPUT
!...
!... Set Tolerances
!...
    ICOR = 80      !predictor corrector term
    E1   = 1.0E-10 !m X space error
    E2   = 1.0E-10 ! Pressure error
    E3   = 1.0E-10 ! Density error
    E4   = 1.0E-10 ! Velocity error
    E5   = 1.0E-10 ! flow angle error
!...
!... Set Control values
!...
    DELTA = 0.0      !1 axi, 0 planer (Mass flux not working correctly)
    NI    = 21       !NUMBER OF RADIAL POINTS ON INFLOW PLANE (Max 99)
    NT    = 11       !NUMBER OF CIRULAR ARC POINTS (Should = AA)
    SMOOTH = .TRUE.  !EVENLY SPACE OUT THE DATA AFTER EACH SECTION
    ST    = 1.0      !0.0 Even space start line, 1.0 quadratic spaced
    IORDER = 25
!...
!... Set Gas Properties
!...
    PS    = 1600.000E+06 !gas pressure
    TS    = 1.000000E+03 !gas temperature
    PA    = 0.0         !ambient pressure
    G     = 1.4         !gamma
    RG    = 287.040     !J/KG-K
    GC    = 1.0         !M-KG/N-S^2
    GL    = 1.0         !M^2/M^2
!...
!... Set Nozzle Properties
!...
    YT    = 1.000 !nozzle radius
    RTU   = 2.000 !radius of curve upstream of throat
    RTD   = 0.500 !radius of curve downstream of throat
    AA    = 15.0 !attachment angle
    AE    = 15.0 !exit angle
    XE    = 25.0 !exit location
!...
!... Set Nozzle Solution
!...
    IVL   = 0         !0 transonic start, 1 User defined input
    EMD   = 3.0       !design mach number
    NOZ   = 0         !1 nozzle shape inputted, 0 nozzle shape from MOC
    I_FINAL = 101    !number of points on final chara line
    BL_SCALE = 1.0   !boundary layer scalar
/

```

APPENDIX I
ERROR MESSAGES

1. Error Code: “**STOPPING CODE: RESET NII TO XXX**”

The variable NII in file “MOC.PAR” needs to be reset to XXX. This error normal occurs if the variable ‘I_FINAL’ in name list \$INPUT is set too large. (Source code: FINAL_CHAR_LINE.f90)

2. Error Code: “**Cannot find namelist &INPUT in file “NOZZLE.inp”**”

The namelist &INPUT was not found in the input file. Make sure that the “&” starts in the second column. (Source code: READ_NAMLIST.f90)

3. Error Code: “**THE RATIO OF RTU/YT IS INVALID FOR SAUERS METHOD!**”

The user input values of RTU and YT are invalid for the code to predict a start line profile. The user must increase the ratio RTU/YT to be greater than 1.0. Ideally this ratio should be greater than 2.0. For values greater than 1 but less than 2, the code will print out a warning message that the ratio is only marginally acceptable. This is a limitation of the use of Sauer’s method^{1,2} for the prediction of the start line profile. (Source code: IVLINE.f90)

4. Error Code: “**File Profile.dat does not exist.**”

The user has set the variable “IVL” to 1 in the namelist there by requesting an inputted start line profile. The file that contains the start line, “Profile.dat,” could not be found within the working directory. (Source code: IVSTART.f90)

5. Error Code: “**Stopping the code: Variable NII needs to be increased.**”

The variable NII in file “MOC.PAR” needs to be increased to a larger value. (Source code: SECOND_ORDER_WALL_FLOWIELD.f90)

6. Error Code: “**ERROR: CAN NOT DECODE FLOW ANGLE!**”

This error is common when the input resolution is not sufficient to resolve the flowfield. To fix this, the user can increase the variable “NI” in the input file. If this error still remains, the user can turn on flow smoothing by setting the logical variable “SMOOTH” to TRUE. (Source code: INTERIOR_POINT.f90 and BCMODS.f90)

7. Error Code: “**Stopping the code: Reached the ICOR user set limit!**”

This error occurs if the variable “ICOR” is set too low or one of the error terms “E1” through “E5” is set too small for the local flow conditions. In general, the variable “ICOR” should be set to 80 and the individual error terms should be raised. (Source code: INTERIOR_POINT.f90)

APPENDIX J
NAMelist INPUT SUMMARY

1 Tolerance Controls

The variables in this section deal with the numerical tolerance controls within the code. In general, these values do not need to be changed.

&INPUT: **ICOR**

Date Type: **Integer**

Description: Specify the number of Predictor/Corrector steps.

Value	Description	Default
> 0	Number of steps	61

- If the “E” tolerances are not being met then this term may need to be increased.

&INPUT: **E1**

Date Type: **Real**

Description: Convergence tolerance for location, m.

Value	Description	Default
> = 0	Minimum error in spatial step error absolute	1.0E-10

- A value of zero means that this term will not be used to determine the convergence of the solution.
- Error is determined by comparing the absolute difference in the position versus the predicted position to the value of E1.

&INPUT: **E2**

Date Type: **Real**

Description: Fractional convergence tolerance for pressure.

Value	Description	Default
> = 0	Minimum fractional error in pressure	1.0E-10

- A value of zero means that this term will not be used to determine the convergence of the solution.
- Error is determined by comparing the absolute difference in the pressure versus predicted pressure to $E2 \times \text{predicted pressure}$.

&INPUT: **E3**

Date Type: **Real**

Description: Fractional convergence tolerance for density.

Value	Description	Default
> = 0	Minimum fractional error in density	1.0E-10

- A value of zero means that this term will not be used to determine the convergence of the solution.
- Error is determined by comparing the absolute difference in the density versus predicted density to $E3 \times \text{predicted density}$.

&INPUT: **E4**

Date Type: **Real**

Description: Fractional convergence tolerance for velocity.

Value	Description	Default
≥ 0	Minimum fractional error in velocity	1.0E-10

- A value of zero means that this term will not be used to determine the convergence of the solution.
- Error is determined by comparing the absolute difference in the velocity versus predicted velocity to $E4 \times \text{predicted velocity}$.

&INPUT: **E5**

Date Type: **Real**

Description: Fractional convergence tolerance for flow angle.

Value	Description	Default
≥ 0	Minimum fractional error in flow angle	1.0E-10

- A value of zero means that this term will not be used to determine the convergence of the solution.
- Error is determined by comparing the absolute difference in the flow angle versus predicted flow angle to $E5 \times \text{predicted flow angle}$.

2 Control Values

The values in this section deal with the solutions resolution.

&INPUT: **DELTA**

Date Type: **Real**

Description: 2-D Axisymmetric or Planar nozzle calculation.

Value	Description	Default
0.0	Planer solution	
1.0	Axisymmetric solution	*

&INPUT: **NI**

Date Type: *Integer*

Description: Specify number of points on the inflow start line.

Value	Description	Default
> 0	Number of points	21

&INPUT: **NT**

Date Type: *Integer*

Description: Specify number of points that define the circular arc wall.

Value	Description	Default
> 0	Number of points	15

&INPUT: **KWRITE**

Date Type: *Integer*

Description: Turns on the output. This should always be set to a value of 1.

Value	Description	Default
1	Output control	1

- Both the output and the errors are directed to FORTRAN unit 6.

&INPUT: **IUNITS**

Date Type: *Integer*

Description: Specify input units. This should always be set to a value of 2.

Value	Description	Default
2	Input Units	2

- All units must be input as standard SI units (N/m, m/s, kg/m³, and so forth).

&INPUT: **ST**

Date Type: *Integer*

Description: Specify radial start line spacing from wall.

Value	Description	Default
0	Even spacing	*
1	Quadratic spacing	

- The user is encouraged to use quadratic spacing as this puts the greatest number of points in the region of greatest change (near wall).

&INPUT: **SMOOTH**

Date Type: **Logical**

Description: Smooth the spacing of points after each marching step.

Value	Description	Default
.F./T.	Smooth the points	.False.

- If true, the solution will be linearly smoothed after each marching step but only AFTER the completion of the initial value start line solution. This option will help prevent points from building up into a strong wave (which may terminate the run). After each step, the code will try to preserve mass along each downward characteristic by modifying the density. All other properties remain the same. This smooth option works best when the variable NI is set to 21 or higher.

&INPUT: **IORDER**

Date Type: **Integer**

Description: Specify the order of the least-squared curve fit for the wall profile.

Value	Description	Default
> 0	Order number	15

- The user may want to experiment with increasing this value to improve the smoothing of the wall profile if strong waves are present. This may be needed if the smoothed wall profile will be used within a full viscous CFD code for verification.

3 Gas Properties

The variables in this section define the reference state of the gas properties of the nozzle flow. This includes the total conditions of the nozzle's chamber, pressure at the nozzle exit, and additional gas constants.

&INPUT: **PS**

Date Type: **Real**

Description: Specify the stagnation gas pressure (N/m) of the nozzle chamber.

Value	Description	Default
> 0	Stagnation Pressure	70.0E+05

&INPUT: PA

Date Type: Real

Description: Specify the ambient back pressure (N/m) at nozzle exit.

Value	Description	Default
>=0	Back Pressure	0.0E+00

&INPUT: TS

Date Type: Real

Description: Specify the stagnation temperature (K) of the nozzle chamber.

Value	Description	Default
> 0	Stagnation Temperature	3.0E+03

&INPUT: G

Date Type: Real

Description: Ratio of specific heats (Gamma).

Value	Description	Default
> 0	Ratio of Specific Heats	1.4E+00

&INPUT: RG

Date Type: Real

Description: Universal gas constant (J/Kg-K).

Value	Description	Default
> 0	Universal Gas Constant	2.8704E+02

&INPUT: GL

Date Type: Real

Description: Unit conversion (M²/M²).

Value	Description	Default
> 0	Unit Conversion	1.0E+00

- This option is not yet fully implemented. Currently the user should not change this default value.

&INPUT: **GC**

Date Type: **Real**

Description: Unit Conversion (M-Kg/N-s²).

Value	Description	Default
> 0	Unit Conversion	1.0E+00

- This option is not yet fully implemented. Currently the user should not change this default value.

&INPUT: **VAR_GAMMA**

Date Type: **Logical**

Description: This option is used for ultra high-pressure air. The local gamma is determined by an internal curve fit as a function of temperature for a large range of pressures. This option should only be used when the working fluid is air.

Value	Description	Default
.F./T.	Variable Gamma for Air	.FALSE.

- The use of this option makes setting the variable G redundant.

4 Nozzle Wall Properties

The variables in this section deal with the initial shape of the nozzle wall. These options define the initial circular arc of the nozzle wall joined tangentially by a second order polynomial.

&INPUT: **YT**

Date Type: **Real**

Description: Nozzle throat radius (m).

Value	Description	Default
> 0	Nozzle Throat Radius	1.0E+00

&INPUT: **RTU**

Date Type: **Real**

Description: Radius of curvature upstream of the throat (m).

Value	Description	Default
> 0	Radius of Curvature Upstream	2.0E+00

&INPUT: **RTD**

Date Type: **Real**

Description: Radius of curve downstream of the throat (m).

Value	Description	Default
> 0	Radius of Curvature Downstream	5.0E-01

&INPUT: **AA**

Date Type: **Real**

Description: Attachment angle (degree) of the last point on the circular arc wall. This is the angle between the circular arc and second order polynomial.

Value	Description	Default
> 0	Wall Attachment Angle	1.0E+01

&INPUT: **AE**

Date Type: **Real**

Description: Exit angle of nozzle assuming no predicted wall.

Value	Description	Default
> 0	Nozzle Exit Angle	1.0E+01

&INPUT: **XE**

Date Type: **Real**

Description: Stopping location of nozzle wall. This option is needed to stop the calculation for a truncated nozzle that does not reach the designed Mach number.

Value	Description	Default
> 0	Stopping Location of Nozzle Wall	1.0E+01

5 Nozzle Solution Parameters

The variables in this section control additional parameters that affect the nozzle flowfield solution.

&INPUT: **EMD**

Date Type: **Real**

Description: Exit design Mach number of the nozzle.

Value	Description	Default
> 0	Exit Design Mach Number	5.0E+00

&INPUT: **BL_SCALE**

Date Type: **Real**

Description: Scale factor on the boundary layer prediction. For nozzles with sudden large expansion ratio after the throat this factor may need to be increased.

Value	Description	Default
> 0	Boundary Layer Scale Factor	1.0E+00

- This value may need to be increase/decreased for very short nozzles of nozzle that have a large expansion angle.

&INPUT: **I_FINAL**

Date Type: **Integer**

Description: Number of points on final characteristic line.

Value	Description	Default
> 0	Number of points	50

- This value may need to be increase if code fails within the turning section of the nozzle. An increase in the term NI should result in an increase in this term.

&INPUT: **NOZ**

Date Type: **Integer**

Description: Nozzle shape.

Value	Description	Default
0	Final nozzle shape will be determined by the MOC code.	*
1	Nozzle shape will be limited to circular arc and second order quadratic wall only	

&INPUT: **IVL**

Date Type: **Integer**

Description: Start line input.

Value	Description	Default
0	Transonic start line	*
1	User inputted start line	

- If $NOZ = 1$, user must supply a "PROFILE.dat" file (see section 2.2.2). This file must start with conditions on the centerline and move up to the wall. It must be in the format of I, J, K, X, Y, Z, Rho, U, V, W, and P; where I, J, and K are integer index points; X, Y, and Z are the point locations (Z is not used); Rho is the density; U, V, and W are the velocity components; and P is the pressure. The temperature is calculated by the equation of state.