1995122324

AN ASSESSMENT OF UNSTRUCTURED GRID TECHNOLOGY FOR TIMELY CFD ANALYSIS

Tom A. Kinard
Lockheed Aeronautical Systems Company
Marietta, Georgia
and
Deanne M. Schabowski
Lockheed Fort Worth Company
Fort Worth, Texas

SUMMARY

An assessment of two unstructured methods is presented in this paper. A tetrahedral unstructured method USM3D, developed at NASA Langley Research Center is compared to a Cartesian unstructured method, SPLITFLOW, developed at Lockheed Fort Worth Company. USM3D is an upwind finite volume solver that accepts grids generated primarily from the Vgrid grid generator. SPLITFLOW combines an unstructured grid generator with an implicit flow solver in one package. Both methods are exercised on three test cases, a wing, and a wing body, and a fully expanded nozzle. The results for the first two runs are included here and compared to the structured grid method TEAM and to available test data. On each test case, the set up procedure are described, including any difficulties that were encountered. Detailed descriptions of the solvers are not included in this paper.

INTRODUCTION

One of the aims of computational fluid dynamics (CFD) is the timely analysis of complete aircraft configurations. To this end, unstructured methods hold considerable promise as a tool by which CFD engineers can efficiently analyze complete aircraft configurations in a timely fashion. Although structured grid methods such as TEAM[1] and CDFALCON[2], based on patched multi-block grids have been applied to complete configurations like the F-22 and F-16, the time to generate such grids remains unacceptably large. In order to reduce the time required to generate grids around complex configurations, unstructured grid technology are being explored. The goal is to reduce turnaround time from weeks to days to hours.

Two unstructured methods are currently being used at Lockheed. The first method, acquired from NASA Langley, is composed of three codes, a grid preprocessor GridTool[3], an advancing front grid generator Vgrid[4], and an Euler flow solver USM3D[5]. The second unstructured method, SPLITFLOW[6], is being developed at the Lockheed Fort Worth Company (LFWC) and uses Cartesian unstructured meshes. This paper compares and contrasts these unstructured methods based on three test cases. The results are compared to available experimental data and to results generated by the patched structured grid method TEAM.

SYMBOLS

C	Chord
C_D	Drag coefficient
C_L	Lift coefficient
C_M	Pitching moment coefficient
C_N	Normal force coefficient
CFL	Courant Friedrichs Lewy number
M	Mach number
x,y,z	Cartesian coordinates
α	Angle of attack

METHOD DESCRIPTION

Tetrahedral Unstructured Method

The tetrahedral grid generation system from NASA Langley is composed of three codes. With these codes, an Euler solution can be generated on simple configurations in a matter of hours.

GridTool. The first code used in the unstructured grid generation process is GridTool. This program takes geometry files in either discrete point or IGES[7] format. Once an adequate geometry file is entered into the program, a user interactively constructs curves and patches on the surfaces exposed to the flowfield. When the entire surface has been divided into patches then the outer boundaries are prescribed, usually with a simple box. Point and line sources are then prescribed, which control the distribution of points not only on the surface but also in the flowfield. A restart option is available to allow the engineer to save intermediate results. This option is particularly helpful in treating complex configurations which may require more than one session to complete the patching. The output of GridTool is an input file for Vgrid. GridTool also has the capability of displaying surface grids on a patch by patch basis, to allow the user to inspect the quality of an unstructured mesh.

Vgrid. The surface and volume grids are generated with Vgrid. Vgrid uses the advancing front method[4] to generate both surface mesh and volume meshes. A structured background[8] mesh is used to define the point distributions for the surface and volume region. The structured background grid is constructed by subdividing the entire flowfield domain into cells. The spacing information for the unstructured grid is stored at the cell nodes. The distributions are determined in a manner similar to the diffusion of heat in a conducting medium from discrete sources.

Once the background grid has been created a surface mesh is constructed by placing points along the edges of the user defined patches, and then triangles are constructed to fill each patch. After each patch is triangulated, the mesh quality is checked automatically and any regions of poor quality are displayed. The user has the ability to change the patch in order to achieve a better meshing if necessary. The surface mesh then forms the initial front for the volume grid. The front is advanced into the field by introducing new points and forming tetrahedra and new faces to complete the grid. This step is usually accomplished in a batch process. The code continues to fill the flowfield domain until either the domain is filled with cells or no more cells can be formed thus leaving pockets or voids in the grid. These pockets are usually filled by removing a layer of cells around the pocket creating a larger void and a new front. The grid generator is restarted and cells are added until the grid is completed. In some cases the background grid has to be modified in order to achieve a complete grid. A grid quality check is then initiated and negative and skewed cells are reported and then corrected by removing cells around the bad cell and refilling. As with the incomplete grid, sometimes the background grid has to be modified in order to remove bad cells.

USM3D. Once an acceptable grid has been generated, the next step is to compute the flow solution using the Euler solver, USM3D. The solver, developed at NASA Langley and solves the time-dependent Euler equations for an ideal gas using a cell-centered finite volume formulation. Spatial discretization is accomplished by the use of Roe's flux difference splitting. The solutions are advanced in time by either an explicit multi-stage Runge-Kutta scheme or an implicit Gauss-Seidel scheme. Local time stepping is used to accelerate the convergence of the solution to a steady state by using a CFL number near the local stability limit. The maximum time step for the explicit scheme is enlarged by the use of implicit residual smoothing. USM3D supports boundary conditions commonly available to Euler solvers. The code is usually run on a Cray-type machine, but can easily be run on other high-end workstations with sufficient memory and computing speed. USM3D uses 44 words per cell of core memory and 26 μ sec per cell per cycle for the explicit version of the code and 180 words per cell of core memory and 64 μ sec per cell per cycle for the implicit version of the code. All computer times are for a Cray YMP.

Cartesian Unstructured Method

SPLITFLOW is an unstructured Cartesian code developed by LFWC for analyzing complex 3-D geometries. SPLITFLOW generates cube-shaped cells that are aligned with the Cartesian coordinate axes. Boundary geometry is defined by triangular faces, or facets. At boundaries, cells are "cut" to account for volume and flux changes due to parts of the cells being inside of the solid surface. This feature allows SPLITFLOW to handle extremely complex geometries, and little care need be taken by the user to prepare or maintain the grid. This type of grid was used on all of the geometries presented in this paper.

Initial grid cell sizes are scaled from geometry facet sizes and are then refined or coarsened, at specified iteration intervals, by the solver based on the user's choice of gradient adaption functions (Mach number, pressure, etc.). The coarsening process uses statistical methods to look for low gradient regions in the flowfield from which to remove cells, thus reducing grid density and computational requirements. The coarsening process is limited by a grid smoothing algorithm which requires adjacent cells to be no more than one "generation" apart (Figure 1). Further, cells are deleted by groups of eight and only if all of the child cells in that group are flagged for coarsening. This is done to maintain the data structure. The refinement process follows, also applying statistical methods, and searches for high gradients to determine where cells need to be added. Grid refinement involves recursively sub-dividing each cell into eight cells which become "children" to the initial cell. Since the code is "smart" enough to place cells where they are needed, the best initial grid is usually sparse and the flowfield is used to determine where new cells should be placed. With a sparse initial grid, flowfield information can propagate in fewer iterations, each of which take less time because there are fewer cells. For example, a grid which is to be limited to 800,000 cells may be appropriately initialized to 50,000 -100,000 cells.

Another benefit of cutting boundary cells is that geometry changes can be made easily while salvaging a developed solution. For example, if the user has a converged solution of an aircraft with undeflected control surfaces, a new geometry model with deflected control surfaces can simply be substituted. SPLITFLOW will recut, or "mark", the appropriate boundary cells and continue solving and refining on the new geometry/flowfield; the cost-effectiveness of such a feature is clear.

SPLITFLOW includes a point implicit solver which typically brings about convergence in under 500 iterations. The amount of memory that is required to run the solver portion of SPLITFLOW is approximately 180 words per cell. The algorithm also includes automatic time step scaling based on the convergence of the sub iterations of the point implicit scheme. The definition has a direct influence on attainable time step size.

WING C TEST CASE

Wing C is a low aspect ratio fighter-type wing. A geometric description of this wing is shown in Figure 2. Wing C was designed to have a large leading edge sweep and mean aerodynamic chord. At the design condition of 0.85 Mach and five degrees angle of attack , the wing has moderate aft loading, mild shocks and mild pressure recovery. The objective of this test case was to evaluate how well each method can model transonic flows with shocks. Extensive force and pressure data has been generated for this geometry on a large scale[9] model and a small scale model[10].

Tetrahedral Unstructured Method Case Analysis

The surface geometry for this test case was generated from tabular airfoil sections and wing characteristic data. A discrete point data file was constructed and used as input to GridTool. Since Wing C has a round leading edge and tip, more patches were needed around these regions to help achieve good resolution. A total of 14 patches were used on the wing surface and outer boundaries. Two line sources were placed at the leading edge and slightly aft of the leading edge in order to achieve adequate resolution in this region. A line source was also placed along the trailing edge and along the chord line at the root and tip of the wing. Eight point sources were placed at the corners of the bounding box to control the spacing in the far fields. The resulting grid contained 47,390 nodes and 266,101 cells. A plot of the upper surface and symmetry plane grid is shown in Figure 3. The cell volumes ranged from a minimum of .1226E-08 near the tip leading edge to .1026E+01 at the outer boundary. The time needed to generate this grid from the initial input geometry was four hours.

With the grid completed, a flow solution was obtained using USM3D at a Mach number of 0.85 and an angle of attack of five degrees. Three boundary conditions were used in this Wing C analysis, a far field, symmetry, and solid boundary condition. The explicit version of the code was used in this test case. The CFL number used for this run was set to four with a smoothing coefficient set to one half. The normal execution procedure for USM3D is to let the code pick when to use low and higher order flux difference splitting. For this case 287 cycles were executed on lower order differencing, or until one order of convergence was achieved. The solution then ran in higher order differencing until 2000 total cycles were reached. At the end of 2000 cycles the residual had reduced by two and a half orders. A plot of the residual convergence is shown in Figure 4. USM3D requires 44 words per cell of memory regardless of what platform chosen to run the code. However, the code performance depends on the platform chosen to run the code. On a Cray YMP the code executes at 26μ seconds per cell per cycle. On a HP 755 workstation the code runs at around 290μ seconds per cell per cycle. For this case, the total solution time was 1.7 Cray C90 hours or 11μ sec per cell per cycle.

Cartesian Unstructured Method Case Analysis

The geometry for the Wing C test case was developed from the surface grid used in the above analysis. The quadrilateral cell database was converted to a faceted surface that was compatible with SPLITFLOW. This conversion process from discrete data to a faceted geometry file required around a half an hour. The facet model contained 10,970 facets. The geometry was a half-wing model with a plane of symmetry. After the faceted geometry has been generated, the time required to start SPLITFLOW is around twenty minutes. The initial grid consisted of 225,281 cells on 18 grid levels. The initial grid size was large due to the extremely curved trailing edge. A plot of the initial grid at the symmetry plane and the upper surface of the wing is shown in Figure 5. The faceted model required a half hour to generate and the input and job file for SPLITFLOW required an additional twenty minutes to prepare.

Starting with this grid, a flow solution was obtained using SPLITFLOW at a Mach number of 0.85 and an angle of attack of five degrees. Three boundary conditions were used in this Wing C analysis: a far field, symmetry and tangent flow for the surface boundary condition. SPLITFLOW has automatic CFL adjustment. In this case the CFL number was limited to 30. The grid adaption occurred every 50 iterations with a final grid consisting of 445,521 cells on 20 grid levels. SPLITFLOW required 4.8 CPU hours on a Cray C90. Grid generation required 0.32 hours out of that total.

Adaptive refinement of the grid was based on gradients of Mach number and pressure. A cutting plane through the 70 percent span location shown in Figure 6 reveals the resolution of the grid near the multiple shocks on the upper surface of the wing.

Solution Comparisons

Figure 7 show comparisons of surface pressures from USM3D, SPLITFLOW, TEAM, and test data. The pressures predicted by TEAM, USM3D, and SPLITFLOW are almost identical except for minor differences near shocks. Both unstructured solutions compare well with the test data along the lower surface and ahead of the shocks. Aft of the shocks there is shock induced separation which Euler codes cannot model. The lack of adequate resolution near the stagnation point is the most likely cause of USM3D not being able to match the stagnation pressure shown in the test data. At the 70 percent span station, the solution from USM3D shows the most difference when compared to TEAM and SPLITFLOW. This is due to inadequate grid resolution at this station. As one moves closer to the tip, the grid used in USM3D becomes finer and yields a better solution, as seen at the 90 percent station. Table I contains a comparison of the computed forces and moments and experimental data. USM3D shows good agreement with TEAM on drag and pitching moment and only slight difference on lift and normal force. USM3D compared well the test normal force. SPLITFLOW predicts a higher drag than either USM3D or TEAM, but is in good agreement on the normal force.

ARROW WING BODY TEST CASE

The second geometry used in this study was an arrow wing body configuration[11]. A schematic of this geometry is shown in Figure 8. The wing on this configuration has a round leading edge and was designed for efficient supersonic cruise. It has both leading and trailing edge flaps. The tip of the arrow wing body configuration is closed by a flat plate. The body has a circular cross-section with a straight centerline. A deflection of 8.3° was imposed on the entire trailing edge flap. The leading edge flaps were kept in the faired or zero deflection configuration. The flow conditions for this case were chosen to be a Mach number of 0.85 and an angle of attack of four degrees. At these conditions the wing is essentially shock free.

Tetrahedral Unstructured Method Case Analysis

The starting geometry for this case was taken from an existing TEAM structured grid. The surface mesh, which consists of discrete point data, was used as input to GridTool. A total of 115 patches were required to completely cover the configuration, sting and far field boundaries. The outer boundaries were located 12 spans from the body. Particular attention was paid to the leading edge, wing-fuselage and wing-flap intersections. Even though care was taken to preserve the rounded character of the leading edge, toward the tip the leading edge of the input geometry becomes relatively sharp, the effect of which is reflected in the unstructured grid solution. Sources were placed along the centerline of the body, wing leading and trailing edges, and along the tip. An additional source was placed just aft of the leading edge in order to help add points in that region. Some difficulties were experienced near the tip where the cell sizes become extremely small compared to the characteristic length of the wing. Although no negative cells were found, there existed some highly skewed cells near the leading edge in the tip region of the grid. The final grid consisted of 57,788 nodes and 307,677 cells. A plot of the surface grid is shown in Figure 9. The cell volumes ranged from 0.246E+07 near the outer boundary to 0.288E-06 near the wing tip leading edge. The set up time for this grid was 24 man hours.

As with Wing C, only symmetry, far and solid boundary conditions were used in this solution. The numerical parameters used in this run were a CFL of 4 and smoothing coefficient of a half. A three stage explicit scheme was used throughout this run. The code switched from lower order differencing to higher order differencing after 181 cycles. The first attempt to run this grid failed shortly after switching to higher order differencing. The pressure and density approached zero in the small cells that were around the wing tip leading edge. A modification to the code which set these cells to first order allowed the code to continue to run. A total of 461 cells were set to first order after 2000 cycles had been executed. After 2000 cycles the residual had reduced by two and a quarter orders. A plot of the residual convergence is shown in Figure 10. The total solution time for this case was 2.5 Cray C-90 hours. or 10μ sec per cell per cycle.

Cartesian Unstructured Method Case Analysis

The geometry for the Arrow Wing Body test case was developed from a baseline IGES file. The model was modified to define a full span 8.3 degree trailing edge deflection. The model contained 34,034 facets defining one half of the aircraft. The initial grid consisted of 27,737 cells.

A SPLITFLOW flow solution was obtained at a Mach number of 0.85 and four degrees angle of attack. Three boundary conditions were used in this Arrow Wing Body analysis: a far field, symmetry and tangent flow for the surface boundary condition. The grid adaption occurred every 40 iterations with a final grid consisting of 169,749 cells. A pressure coefficient contour plot of the symmetry plane grid and surface geometry is shown in Figure 11. In this case, the CFL was limited to 8.0. SPLITFLOW required 0.69 CPU hours on a Cray C90. Grid generation required 0.11 of that total.

Although the solution was run to 197 iterations, the data shown in Figure 12 shows that the force and moment data were converged within engineering accuracy (assuming a data uncertainty band of +/- 0.05) at approximately 100 iterations. The grid at that point consisted of 60,000 cells. The CPU time at 100 iterations was 0.24 Cray C90 hours.

Solution Comparisons

Figure 13 show comparisons of surface pressure coefficients from USM3D, TEAM, SPLITFLOW and test data. The effect of the flap defection can be seen at the inboard span station. The leading edge peak suction pressure from USM3D agrees better with the test data than the TEAM results, due to insufficient grid resolution in the TEAM grid. USM3D and SPLITFLOW agree well except at the 80 percent span station. The USM3D pressures generally agree with the test data except aft of the flap deflection. The SPLITFLOW pressures show similar behavior to the TEAM results in the region aft of the flap break. At the tip the results are less encouraging. In this region there is a small vortex that is not picked up by any of the codes due to the inviscid nature of the computations. The peak suction predicted by USM3D is much higher than TEAM or SPLITFLOW. Both TEAM and USM3D agree with the test data on the lower surface. On the upper surface aft of the first 10 percent, both USM3D and TEAM show similar behavior. The SPLITFLOW results show good agreement at the leading edge but show less agreement with test data or TEAM results especially near the trailing edge. Table II shows a comparison of computed forces and moments for all codes and test data. Only normal force and pitching moment numbers were available in the wind tunnel report. The normal force predicted by USM3D is higher than the test data and the normal force computed by TEAM. SPLITFLOW shows good agreement with TEAM on lift and normal coefficient. SPLITFLOW predicts a higher drag and lower pitching moment than either USM3D or TEAM.

NASA 2D-CD NOZZLE TEST CASE

The objective of this test case was to assess each code's ability to correctly model flows internal and external to a 2D-CD[12] nozzle. A schematic of the nozzle chosen is shown in Figure 14. The nose-forebody section of the model followed a smooth external transition from a circular cross section at the conical nose to a superelliptical cross section at fuselage 26.5. The maximum external cross-sectional area of 41.17 in^2 occurs at fuselage station 26.5. The cross-sectional area and the external geometry remained constant from fuselage station 26.5 to fuselage station 55.05. For this study only the section from fuselage 26.5 to the exit of the geometry was modeled. The nozzle geometry has straight, parallel internal sidewalls. The nozzle-to-throat area ratio of 1.25 yielded a design exit Mach number of 1.6 and nozzle pressure ratio (ratio of the local total pressure to the free stream static pressure) of 4.25. Extensive tests were done on this model at NASA, but only one case will be examined here. An attached external flow test case was chosen for this study. The free stream Mach number was 0.6 zero degrees angle of attack, and a nozzle pressure ratio of 4.0.

At the time this paper was written, (February 1995) results from both SPLITFLOW and USM3D were not available. Grids had been generated, but flow solutions were proving to be a challenging undertaking. Both USM3D and SPLITFLOW are still under development and appropriate boundary conditions for modeling propulsion type flows are being investigated.

CONCLUDING REMARKS

The use of unstructured methods can substantially reduce the time required to generate Euler solutions on complex configurations as compared to structured grid methods. Two unstructured methods were applied to three test cases to assess their strengths and weaknesses. The tetrahedral method USM3D showed good agreement and robustness on the two external flow test cases. SPLITFLOW, the Cartesian unstructured method, converged in fewer cycles than USM3D and showed the benefits of adaptive grid refinement.

The set up time for the two codes showed the most difference. SPLITFLOW required the least amount of time for problem set up. This is due to the fact that the grid generator is an integral part of the solver. Only the surface mesh needs to be input to the code. If IGES geometry is available to the engineer constructing the SPLITFLOW input files, the time to set up a problem is on the order of a half an hour. If discrete point data is used then the set up time can run into a number of hours. This is in contrast to USM3D which requires the full volume grid to be input to the solver. Depending on the complexity of the geometry this grid generation time can run from a few hours to a few days. For wing alone cases, the grid generation time is on the order of hours. For full configuration aircraft, the grid generation time can be on the order of a few days.

When the solvers are examined, comparisons are harder to arrive at. The memory needed to run SPLITFLOW is on the order of 180 words per cell. This can vary depending on the way the code is initialized. For USM3D, the memory is easier to determine since the grid generator is a separate code. Using the explicit option in USM3D, the memory required for execution is 44 words per cell, using the implicit option the memory increases to 180 words per cell. Since grid generation time is an integral part of SPLITFLOW the execution time in terms of time per cell per cycle is difficult to quantify.

A converged solution is achieved in USM3D when two orders of residual reduction and the forces and moments have stopped oscillating. A similar situation exists for SPLITFLOW, but the primary convergence criteria is forces and moments. Solutions are stopped when the forces and moments have reached a steady state.

On a wing alone test case both unstructured methods showed similar results. Both methods were shown to have the ability to model external flowfields with good accuracy. Further development of the solvers is required to treat the nozzle test case representative of a combined external / internal flow-field.

REFERENCES

1. Goble, B.D., Raj, P., and Kinard, T.A., "Three-Dimensional Euler/Navier-Stokes Aerodynamic Method (TEAM), Version 713 User's Manual," WL-TR-93-3115, February 1994.

2. Reed, C.L., "Central Difference Falcon User's Manual," June 1994.

3. Abolhassani, J.S., "Unstructured Grids on Nurbs Surfaces", AIAA-93-3454, 1993.

4. Parikh, P., Pirzadeh, S., and Löhner, R., "A Package for 3-D Unstructured Grid Generation, Finite-Element Flow Solution, and Flow Field visualization," NASA CR-182090, September 1990.

5. Frink, N.T., Parikh, P., and Pirzedeh, S., "A Fast Upwind Solver for the Euler Equations on 3-D Unstructured Meshes," AIAA-91-0102, 1991.

6. Karman, S. L., "SPLITFLOW: A 3d Unstructured Cartesian/Prismatic Grid CFD Code For Complex Geometries", AIAA-95-0343, January 1995

7. "The Initial Graphics Exchange Specification (IGES) Version 5.0," Distributed by National Computer Association, Administrator, IGES/PDES Organization, 2722 Merrilee Drive, Suite 200, Fairfax, VA 22031.

8. Pirzadeh, S., "Structured Background Grids for Generation of Unstructured Grids by Advancing Front Method," AIAA-91-3233, 1991.

9. Keener, E.R., "Pressure-Distribution Measurement on a Transonic Low-Aspect Ratio Wing," NASA TM 86683, September 1985.

10. Hinson, B.L., and Burdges, K.P., "Acquisition and Application of Transonic Wing and Far-Field Test Data for Three Dimensional Computational Method Evaluation," AFSOR-TR-80-0421, March, 1980.

11. Manro, M.E., Manning, K.J.R., Hallstaff, T.H. and Rogers, J.T., "Transonic Pressure Measurements and Comparison of Theory to Experiment for Arrow-Wing Configuration", Volume II: Experimental Data Report-Effects of Control Surface Deflection,NASA CR-132728, October 1975

12. Compton III, W.B., Thomas, J.L., Abeyounis, W.K., and Mason, M.L., "Transonic Navier-Stokes Solutions of Three-Dimensional Afterbody Flows," NASA TM 4111, July, 1989.

Table I. Forces and Moments for Wing C

Wing C Mach = 0.85 $\alpha = 5.0°$				
	CL	CD	CN	CM
USM3D	0.5407	0.03854	0.5420	-0.07792
SPLITFLOW	0.5374	0.04026	0.5389	-0.06813
TEAM	0.5650	0.03995	0.5653	-0.07782
TEST			0.540	

Table II. Forces and Moments for Arrow Wing Body

Arrow Wing Body Mach = 0.85 $\alpha = 4.0°$				
	CL	CD	CN	CM
USM3D	0.2748	0.01747	0.2812	-0.1253
SPLITFLOW	0.2610	0.02484	0.2683	-0.0857
TEAM	0.2666	0.02071	0.2674	-0.0947
TEST			0.276	-0.106

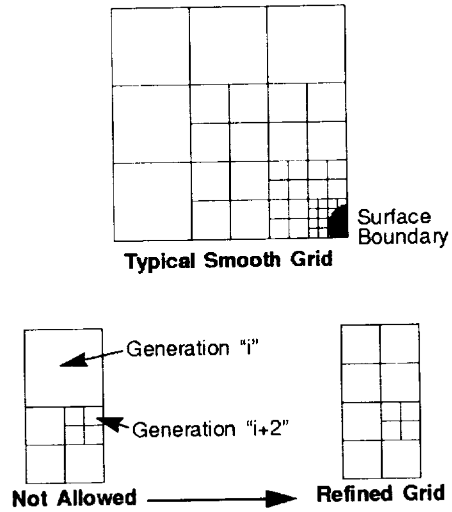


Figure 1. SPLITFLOW cell development

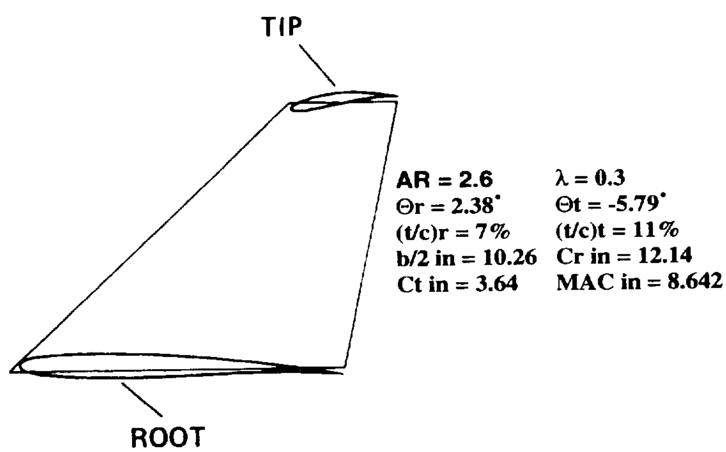


Figure 2. Schematic representation of Wing C

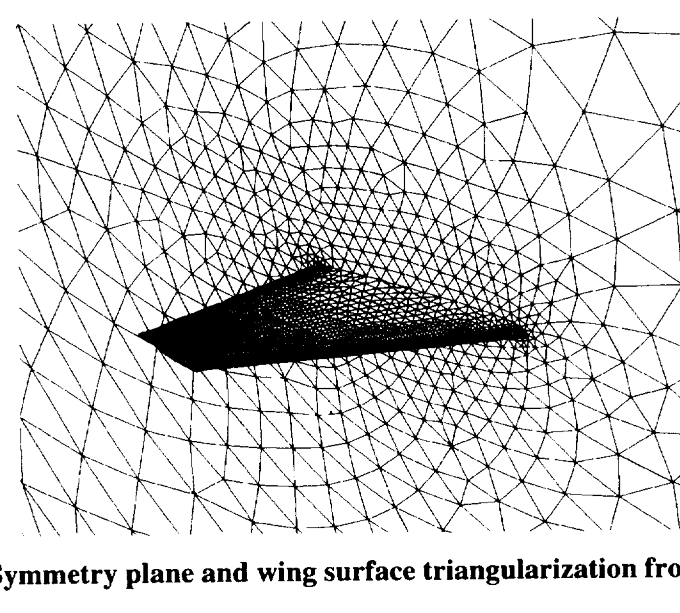


Figure 3. Symmetry plane and wing surface triangularization from Vgrid

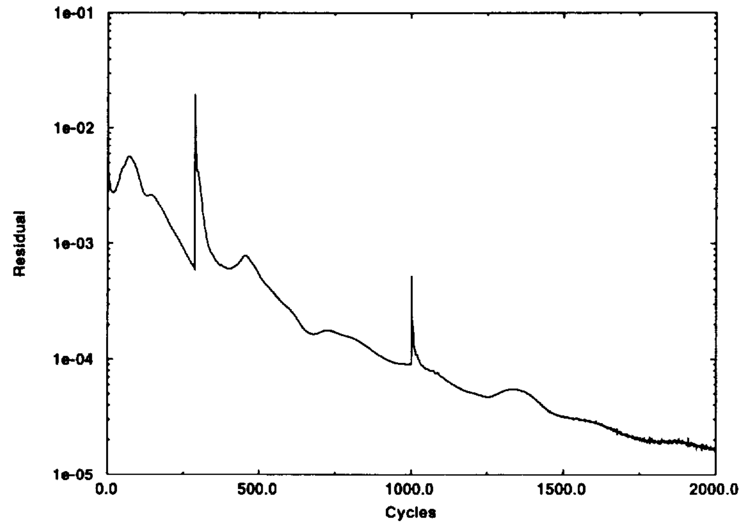


Figure 4. Residual history from USM3D for Wing C test case

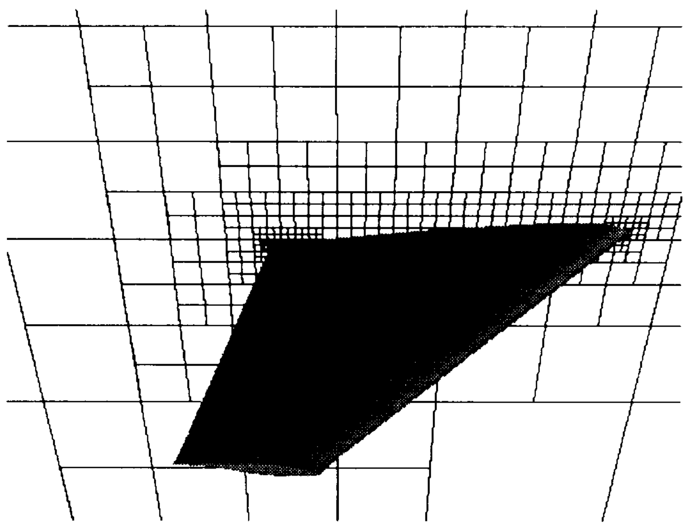


Figure 5. Initial symmetry plane grid from SPLITFLOW

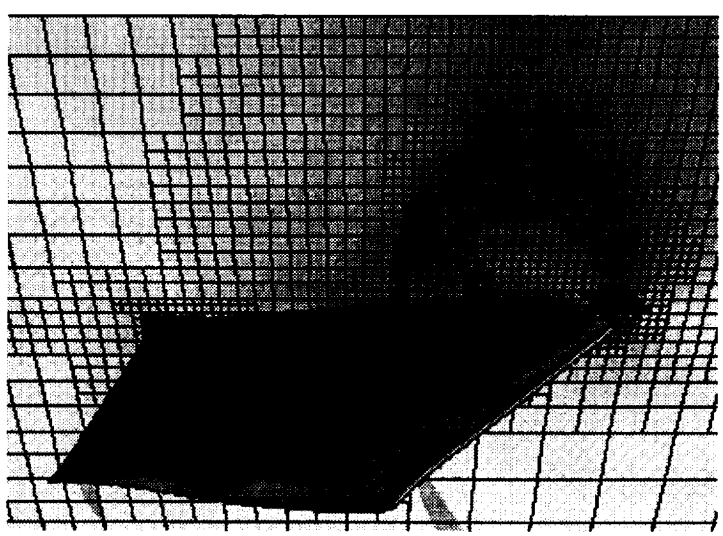
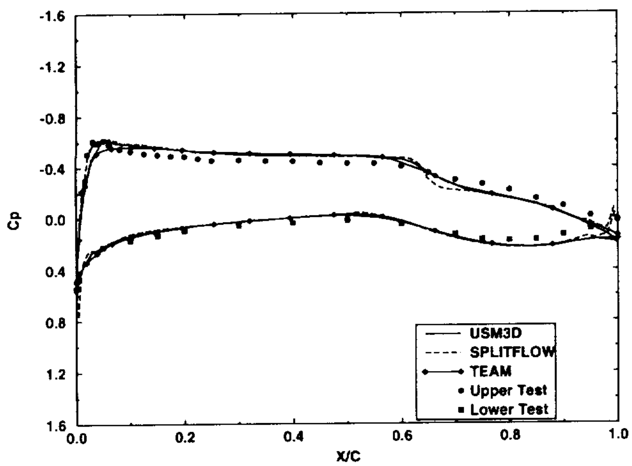
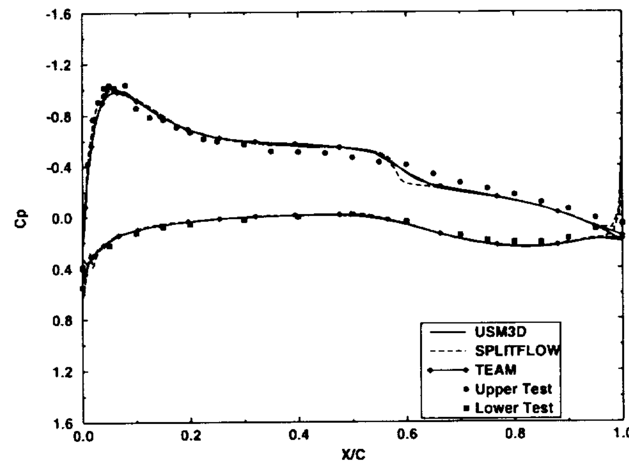


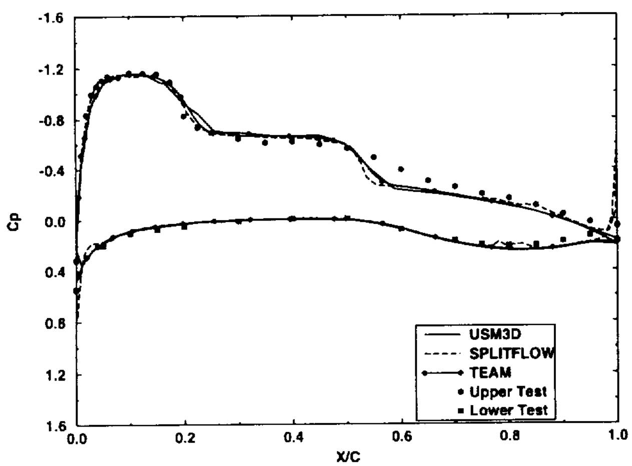
Figure 6. Grid and solution from SPLITFLOW at the 70% span station



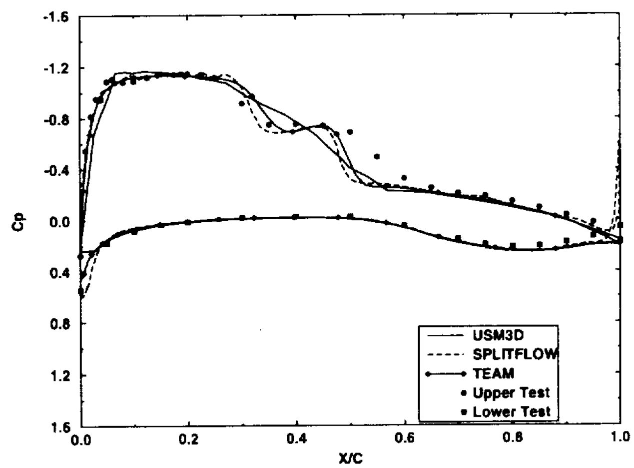
Eta = 0.1



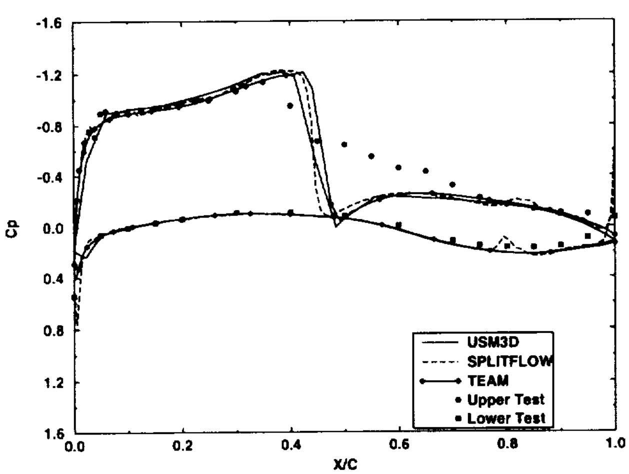
Eta = 0.3



Eta = 0.5



Eta = 0.7



Eta = 0.9

Figure 7. Pressure distributions on Wing C at $M_\infty = 0.85$ and $\alpha = 5.0°$ using USM3D, SPLITFLOW, and TEAM compared to test data

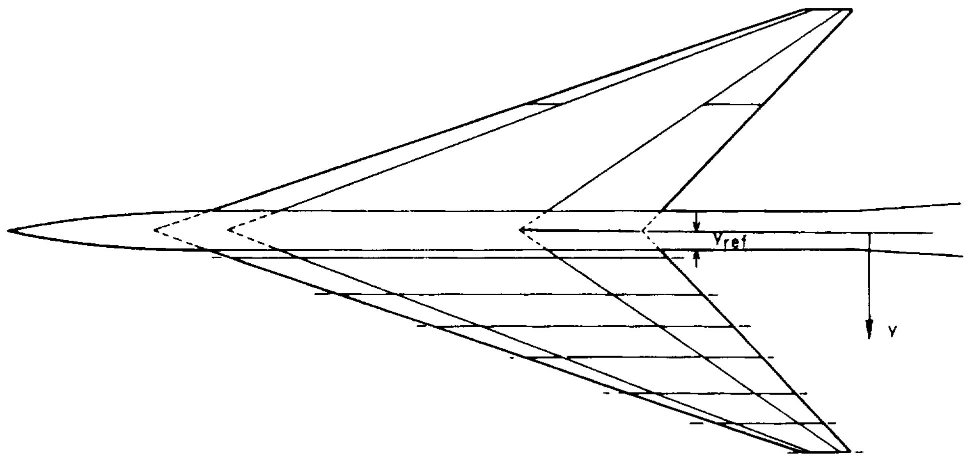


Figure 8. Schematic representation of Arrow Wing Body configuration

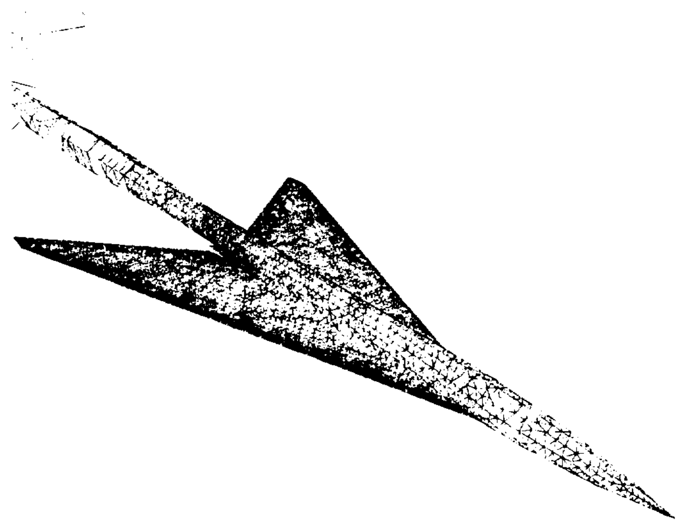


Figure 9. Surface grid on Arrow Wing Body configuration from Vgrid

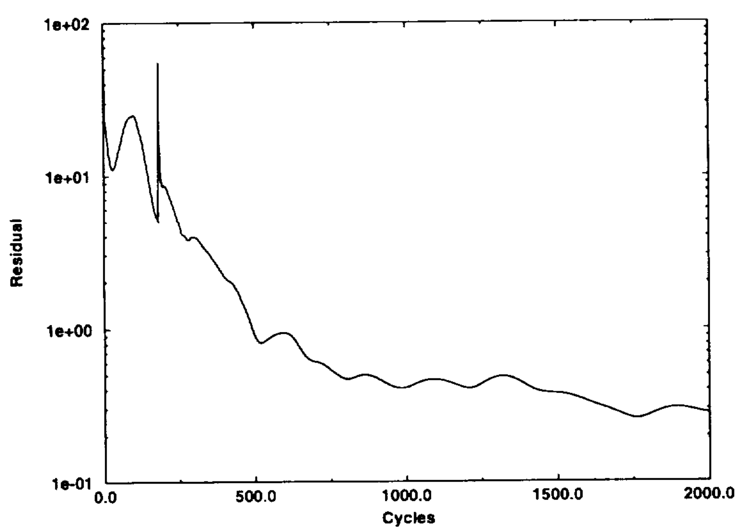


Figure 10. Residual history from USM3D on Arrow Wing Body

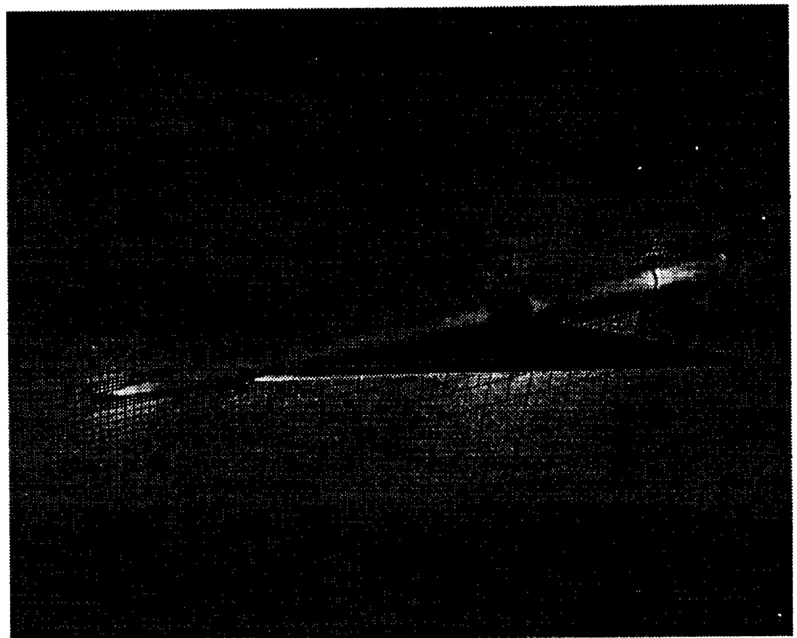


Figure 11. Pressure Contours from SPLITFLOW for Arrow Wing Body case at $M_\infty = 0.85$ $\alpha = 4.0$ and $\delta_{TEF} = 8.3°$

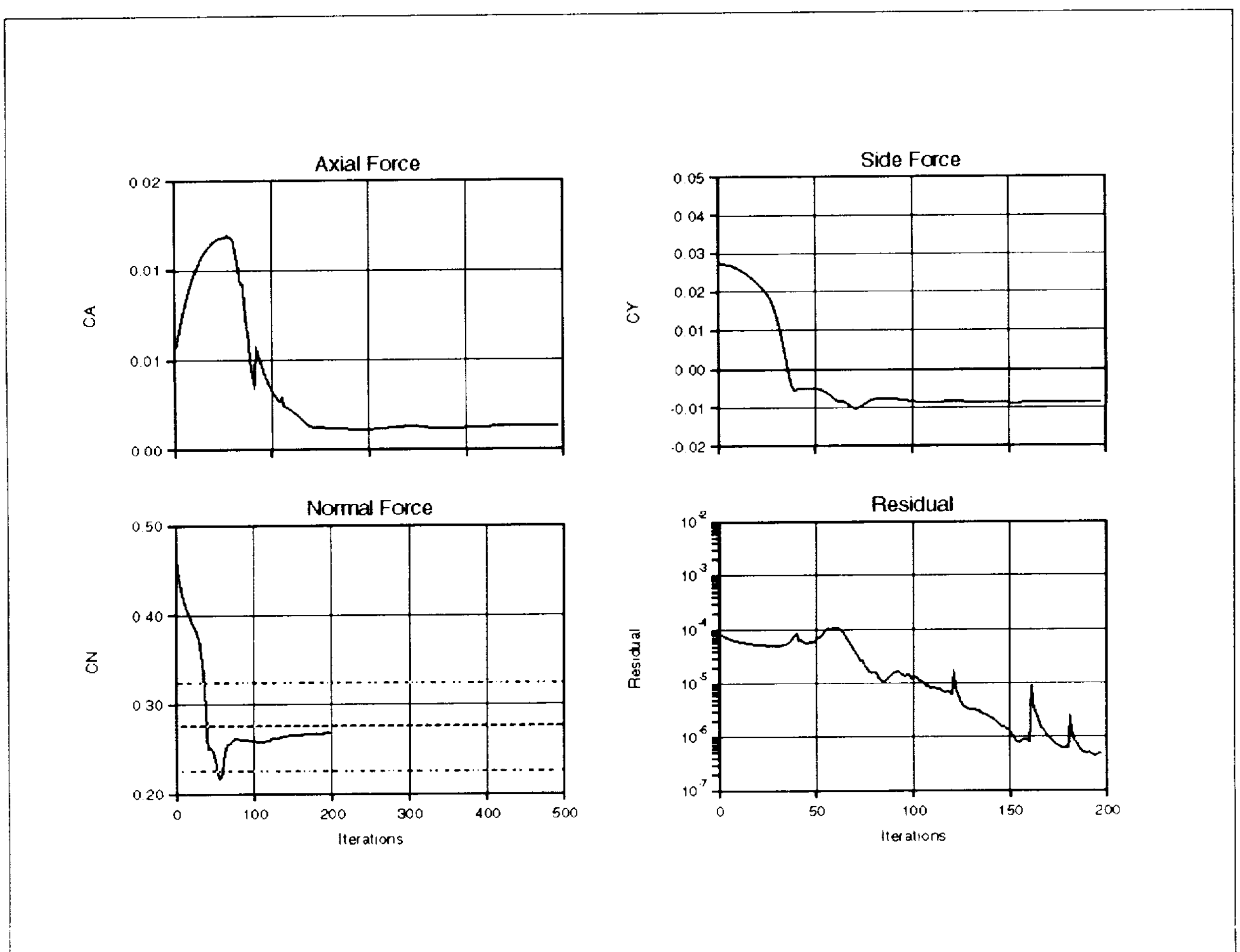
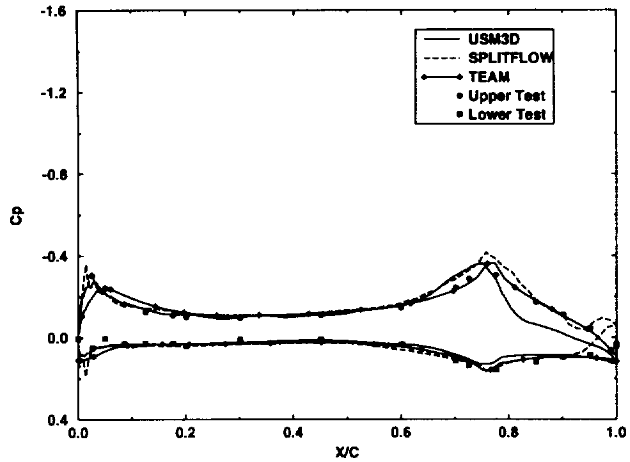
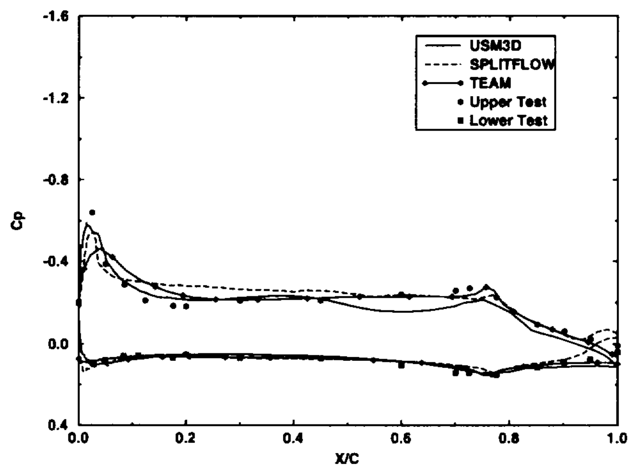


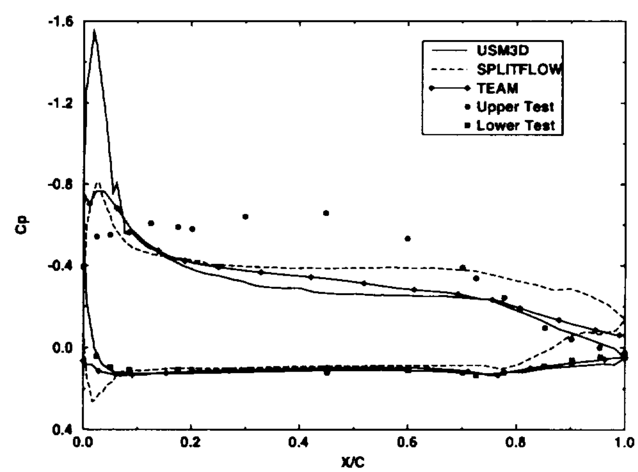
Figure 12. Convergence data from SPLITFLOW for Arrow Wing Body case at $M_\infty = 0.85$ $\alpha = 4.0$ and $\delta_{TEF} = 8.3°$



Eta = 0.2



Eta = 0.5



Eta = 0.8

Figure 13. Pressure distributions on Arrow Wing Body at $M_\infty = 0.85$ and $\alpha = 4.0°$ using USM3D, SPLITFLOW, and TEAM compared to test data

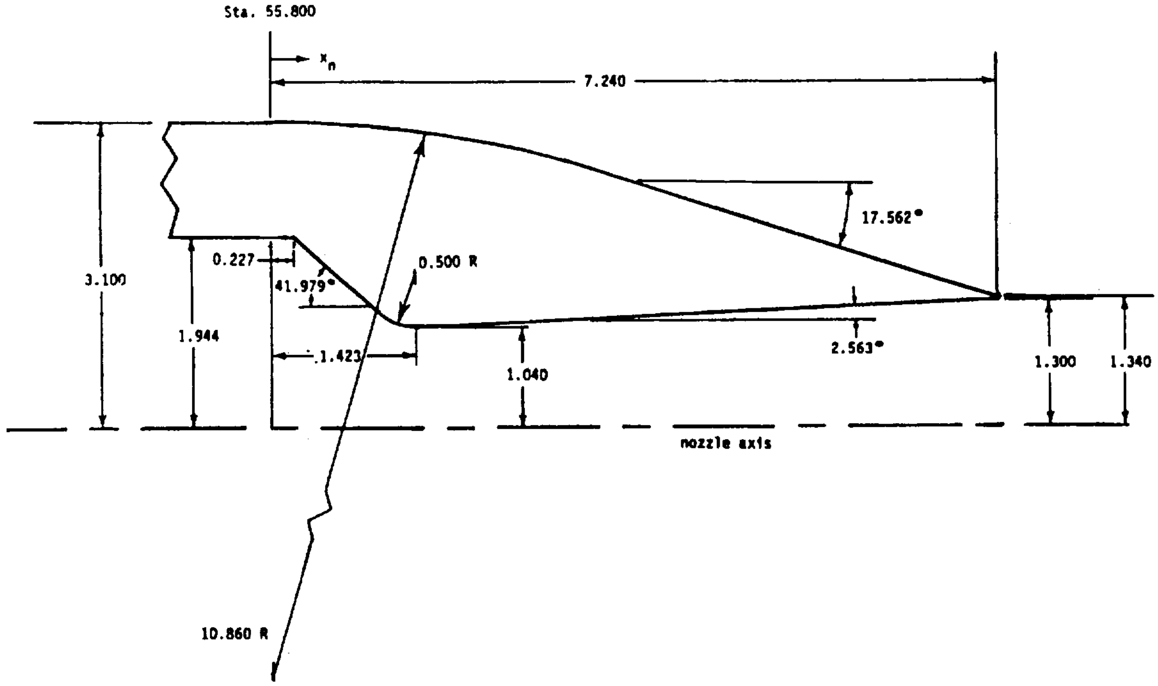


Figure 14a. Top view schematic of NASA 2D-CD Nozzle

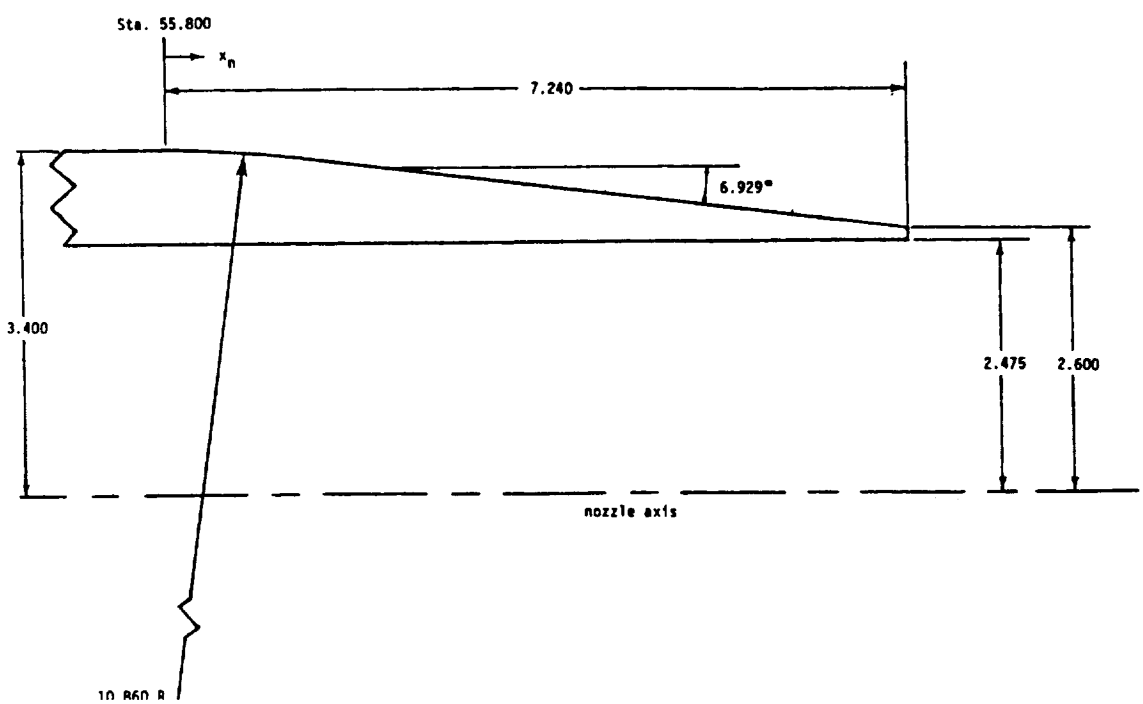


Figure 14b. Side view schematic of NASA 2D-CD Nozzle