ARMY RESEARCH LABORATORY

Computational Fluid Dynamics Modeling of Parachute Clusters

Jubaraj Sahu
Richard J. Benney
Sekaripuram V. Ramakrishnan

ARL-TR-1440

NOVEMBER 1997

Army Research Laboratory
Aberdeen Proving Ground, MD 21005-5066

ARL-TR-1440 November 1997

Computational Fluid Dynamics Modeling of Parachute Clusters

Jubaraj Sahu
Weapons & Materials Research Directorate, ARL

Richard J. Benney
Natick Research, Development, and Engineering Center
U.S. Army Soldier Systems Command

Sekaripuram V. Ramakrishnan
Rockwell International Science Center

Abstract

A computational study has been performed to determine the aerodynamics of a cluster of round parachutes using computational fluid dynamics (CFD). The results given here are the first predictions of descent characteristics for a cluster of three half-scale C-9 parachutes. In particular, the results include the aerodynamic flow field and geometry of the parachute cluster with assumed shapes for the individual canopies. Computed results have been obtained using both structured and unstructured numerical techniques. The computed pressure over the inner and outer surfaces of a single canopy in the cluster is used to calculate the net forces and moments acting on the canopy. A manual iterative process is used to determine the expected stable configuration for the cluster geometry by determining the condition at which the net moment about the payload (origin) is zero. The corresponding angular location of the canopies predicted by the CFD computations is compared with the available experimental results and is found to agree well with the data. It is shown that significant progress has been made in determining the terminal descent flow field characteristics of a particular parachute cluster configuration. These computational solutions provide, for the first time, an understanding of the flow field in and around parachute clusters.

ACKNOWLEDGMENTS

The authors are indebted to Dr. K. Y. Szema, Mr. Kung Lee, and Mr. Dale Ota of the Rockwell International Science Center for their numerous technical discussions of and help with the parachute cluster problem, especially with the grid generation.

INTENTIONALLY LEFT BLANK

TABLE OF CONTENTS

INTENTIONALLY LEFT BLANK

LIST OF FIGURES

INTENTIONALLY LEFT BLANK

COMPUTATIONAL FLUID DYNAMICS MODELING OF PARACHUTE CLUSTERS

1. INTRODUCTION

Parachute clusters have been used in a wide range of applications throughout their history, and the process utilized for their development and design has significantly improved over the years. Parachute cluster systems were developed to avoid the use of excessively large single canopies. A practical limit for the utility of single round canopies is approached, as the users' desire for recovering larger payload weights increases. In many applications, a cluster of parachutes has a number of benefits over a large single canopy. These include the ability to rig and manufacture the system. Clusters also have the advantage of backup protection. This is evident when a single cluster chute failure is considered. The system under a smaller subset of the canopies still has a lower impact velocity than a free fall. Clustered parachute systems have excellent stability characteristics compared to most single-canopy systems. This is partly because of the terminal descent shape of multiple bluff bodies versus a single bluff body. Of course, there are some undesirable features of parachute clusters, which include the difficulty in obtaining a time-sequenced opening of all canopies together. This phenomenon is known as "lead lag," in which one canopy opens quicker than the next canopy. This generally forces the individual parachutes in the cluster to be stronger than would be needed for a consistent opening of all canopies together. Another disadvantage of parachute clusters is that the total drag from each canopy in the cluster is less than the drag that would be experienced by the equivalent single canopy at the same descent velocity. This reduction in drag is attributable to the shape about which clustered parachute systems oscillate. Each canopy in the cluster is most often positioned at some angle of attack to the trajectory path of the system, which reduces the drag produced by each canopy. The optimization of a cluster configuration can result in a trade-off between the overall height of the system, which affects the opening time, etc., to the lengths used for the risers.

Most cluster applications described in the literature are National Aeronautics and Space Administration (NASA) systems, including the Apollo recovery systems and the space shuttle solid rocket booster recovery systems.[1-6] However, parachute clusters are employed for many other applications, including numerous military applications. Military systems that use clusters include extraction systems and low-velocity airdrop systems (LVADS)[7] for cargo, which consistently deliver payloads as heavy as 60,000 pounds (see Figure 1). The G-11 and G-12 are the most common military parachutes that are used for cargo platforms in a variety of configurations based on the payload weight. For example, a 60,000-pound system uses a cluster of three 28-foot-diameter extraction canopies that extract twelve 100-foot-diameter G-11 canopies. Lower weights use fewer canopies such as a 20,000-pound system, which typically uses a single 28-foot ring slot extraction canopy to extract four 100-foot-diameter G-11 canopies. One design challenge of these systems is the desire for commonality. The users want systems that can be employed for a wide range of weights; they also want to minimize the number of separate items needed in inventory. Therefore, it is desirable to have a predictive tool that can be used to accurately predict the terminal descent velocity, configuration, and stability of a given system for a wide range of payload weights. A computational tool that models the terminal descent characteristics of a cluster of parachutes is a technology that is needed by parachute designers and engineers. A joint effort between the U.S. Army Soldier Systems Command, Natick Research Development and Engineering Center (Natick), and the U.S. Army Research Laboratory (ARL) to develop this computational tool is now under way. Current cluster parachute systems are overly designed and often poorly optimized because the interacting flow field associated with parachute clusters is not understood.

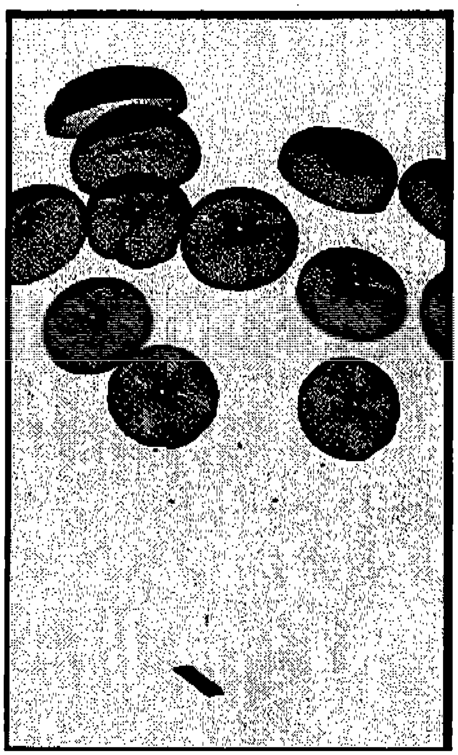

Figure 1. LVAD Parachute Cluster.

As with most parachute systems, the tools available to predict performance and to design a clustered parachute system are based on empirical data and comparison with proven systems. A lot of experience helps. Some fundamental experimental work has been done with clustered parachute systems.[8,9] However, very little is understood about the flow field characteristics in and around clustered parachutes. One goal of this work is to provide a fundamental understanding of parachute cluster flow fields. The ability to model the opening process of a cluster of parachutes is extremely complex and is not a current goal of this work. The goal of this work is to model terminal descent characteristics and the stability of the system once all canopies are open. The ultimate goal is to develop a predictive tool that can be used to optimize a parachute cluster's terminal descent characteristics. This will include the ability to predict the effect of modifying riser lengths and suspension line lengths on the canopies.

The ability to accurately simulate the flow field around a parachute cluster is a complex problem. The aerodynamic characteristics associated with a single or cluster of parachutes in the terminal descent phase is extremely complex to model. The complexity arises largely from the fact that the flow field depends on the canopy shape, which itself depends on the flow field. A correct model must include the coupled behavior of the parachute system's structural dynamics with the aerodynamics of the surrounding flow field. A coupled model is required to determine the terminal descent characteristics of clustered parachutes including velocity, shape, drag, pressure distribution, and the other flow field characteristics. As a starting point, the present research has focused on the use of computational fluid dynamics (CFD) to gain a basic understanding of the interference aerodynamic flow fields associated with parachute clusters. Prior work included CFD modeling of single "axisymmetric" and three-dimensional (3-D) canopies. In this case, CFD techniques were used to model a fixed shape single canopy in 3-D. The results were manually coupled to a static structural code that predicted the canopy shape based on a CFD-supplied canopy surface pressure distribution.[10,11] These solutions also provided a separately produced set of flow field predictions from a completely different CFD software tool. Computed results were compared with numerical results, which were obtained earlier and used with other concurrently developed single-canopy opening models. For CFD modeling of parachute clusters, we have chosen to model a cluster of three half scale C-9 flat circular parachutes. The rationale for choosing this configuration was our ability to compare some aspects of the solutions to recently completed experimental studies of this cluster configuration during controlled conditions by Lee, Lanza, and Buckley.[8] Once the model is validated against these experiments, the procedure can be applied to other cluster systems. This report describes the approach chosen to model the cluster configuration, the gridding procedure required, the solution scheme, the method of comparison to the experimental work, and planned future direction.

2. SOLUTION TECHNIQUE

2.1 Approach

As with our single-canopy models, it is assumed that the shape and configuration of the cluster canopies are known. The initial surface shapes used for each canopy (axisymmetric about each canopy in the current model) are the steady state shapes determined from previous solutions of the single-canopy model falling at approximately (within ±1 ft/sec) the same terminal descent velocity.[10] The symmetric shape of each canopy is an approximation, and the accuracy of this assumption will be studied by using a 3-D structural finite element program[12] by feeding a full 3-D representation of the canopy with the CFD predicted surface pressure distribution in the near future. It is anticipated that a few iterations will be necessary to converge on the correct canopy shape as was done previously with the single-canopy models.[11]

A canopy surface grid (for a given cluster configuration) at a prescribed polar angle (defined as ϕ) is generated by using a Formula Translator (FORTRAN) code to rotate each node point of the single parachute geometry in 3-D about a user-defined confluence point. The polar angle, ϕ, is measured from the vertical y-axis with the xz plane being parallel to the ground. This fully defines the shape of the system being modeled. To modify suspension line lengths, a new single-canopy geometry must be generated. Then the riser lengths can be modified by moving the payload confluence point and rotating the geometry about the vertical axis. For the present problem, four separate canopy surface orientations were considered and a full 3-D CFD grid was generated for each case. For each configuration, the CFD code was run with the same boundary conditions and the same user-supplied in-flow velocity (which is the experimentally determined system's terminal descent velocity). This in-flow velocity is defined to be the velocity that was measured from the previously referenced experiments. Solutions for each of the four configurations were obtained, and the canopy surface pressure distributions were extracted from the solution output. Note that the final solution is a function of the in-flow velocity, which ultimately defines the payload weight for the model.

The results from each of the configurations were used to determine the correct configuration (polar angle for each canopy) for a given cluster system. The canopy surface pressure distribution predicted by the CFD solution is extracted from the overall 3-D volume solution. A summation procedure is used to calculate the net forces on the fixed canopy. The same numerical fluxes used in the code for the parachute surfaces (inner and outer) are used in the force and moment computations. The area of each triangle is multiplied by the pressure at the cell center to determine the net force acting on each triangular cell. The forces and moments are summed over all interior and exterior canopy triangular cells. The result of this calculation yields the net forces and moments acting on the canopy. The moment calculation is performed to determine the tendency of the cluster configuration to rotate toward or away from the system's vertical y-axis. If the configuration consists of canopies that are too close to the vertical axis, the summation of moments produces a net desire for the configuration to separate from the vertical axis. A plot of the net moment for each configuration as a function of ϕ (the polar angle for each canopy in the cluster) then yields the predicted steady state configuration for the system at the prescribed in-flow velocity. This is approximated as the value of ϕ at which the net moment is zero. This value of ϕ is the computed steady state terminal descent polar angle about which the system oscillates. Concurrently, the net vertical drag produced by the system is determined as the net vertical force of one canopy multiplied by the number of canopies. This net force is the computed payload weight that will descend at the terminal velocity used in the CFD calculation. Therefore, by repeating this process for a variety of in-flow velocities, a complete understanding of the system for a range of payload weights can be determined. Note that the CFD solutions do not reach a completely constant velocity and pressure field because of vortex shedding, etc. Therefore, it may be necessary to repeat these calculations over a period of time once the CFD solution has evolved.

2.2 Computational Fluid Dynamics Models

The computational method used in the initial analysis solves the incompressible Navier-Stokes equations in 3-D[13] generalized coordinates for low-speed flows. This technique is based on the method of artificial compressibility and an upwind differencing scheme. The pseudo-compressibility algorithm couples the pressure and velocity fields at the same time level and produces a hyperbolic system of equations. The upwind differencing leads to a more diagonal system and does not require a user-specified artificial dissipation. The viscous flux derivatives are computed using central differencing. This code is capable of computing both steady state and time accurate flow fields. The method is a finite volume implementation of high accuracy upwind schemes embedded in a multizone, structured grid framework.

The governing equations are numerically represented and solved using a nonfactored Gauss-Seidal line-relaxation scheme. This maintains the stability and allows a large pseudo-time step to be taken to obtain steady state results. Details of the numerical method are given in Pan and Chakravarthy.[13] For computation of turbulent flows, a turbulence model must be specified. The structured flow solver uses a two-equation k-ε turbulence model. In this model, two transport equations are solved for the turbulent kinetic energy and the turbulent dissipation rate. It is solved using the same Gauss-Seidal-type line-relaxation scheme used to solve the mean flow equations.

Another evolving technology that has high potential in reducing the preprocessing (grid generation) time is unstructured methodology. A recently developed unstructured flow solver[14] by Rockwell Science Center is used in the present parachute cluster problem. It uses a generalized Lax-Wendroff scheme for solving Euler equations on unstructured grids. This is a compact second order scheme that employs upwinding even in the computation of the gradients of the dependent variables. The generalized Lax-Wendroff scheme is less involved, much more compact, and has the potential to be a good scheme for a wide range of flows. Also, the compactness of the scheme enhances its stability and enables use of larger values for the time step.

This approach employs a multilevel time-stepping scheme. The second order scheme uses a two-time-level discretization. For the first fractional step, q_L and q_R are set equal to the corresponding cell-center values. Here, q contains all dependent flow variables. Roe's approximate Riemann solver is used to obtain a single value for q $(=q^*)$ at the center of a cell. For the second fractional step, q_L and q_R (left and right states) are obtained from a Taylor series expansion of q about the centroid of the corresponding cell. The algorithm just described can be considered as a generalization of Lax-Wendroff upwind integration, since it reduces to the Lax-Wendroff scheme for uniform rectangular hexahedral cells. The solver does not require any information regarding the shapes of the cells and the faces. This property of the solver renders it suitable for multigrid techniques. In order to be able to compete with integral methods that are generally employed in the design process, a multigrid version of the scheme described was developed to improve the convergence rate. This multigrid version of the code has been used in the present parachute CFD modeling.

3. MODEL GEOMETRY AND COMPUTATIONAL GRID

The CFD model was used to determine the 3-D terminal descent characteristics of a cluster of three C-9 canopies. The constructed diameter is 14 feet and suspension lines are 12 feet long. The single half-scale C-9 canopy has been used in a variety of experiments and its opening behavior was predicted with an axisymmetric coupled model developed at Natick.[10] The present modeling effort extends to a cluster of three parachutes.

The terminal descent shape from the axisymmetric model[10] was used to define the surface of the canopies. Because of the symmetry of this problem, a computational mesh was generated around one of the three canopies to reduce the required computer time and memory for the numerical computations. An expanded view of the 3-D grid around the parachute cluster is shown in Figure 2. Also included are the shaded parachute body surfaces. The full grid consists of 12 zones or blocks. It is extremely difficult to generate the grid for this case using one or a very few zones. The full grid consists of a total of 84,690 grid points. The grid points are clustered near the body surface for viscous turbulent flow computations. The outer, in-flow, and down-stream boundaries are placed sufficiently away from the body surface that they do not interfere with the convergence and accuracy of the computed flow field results. Since it is rather time consuming to obtain a structured grid for this problem, one has to assess the use of other alternate grid generation methods, such as the unstructured grid scheme and the Chimera overset grid technique for structured grids. The unstructured grid generation takes considerably less time and was assessed first. The surface grids, both structured and unstructured, of the parachutes in a cluster are shown in Figure 3. After the surface grids are obtained, the outer boundaries are defined, and the 3-D volume grids are generated. The grid generation process is easier with the unstructured grids. Therefore, very little effort would be required to modify the parachute cluster geometry and use the technique to predict cluster configurations of 2,3,4 or more canopies of any diameter.

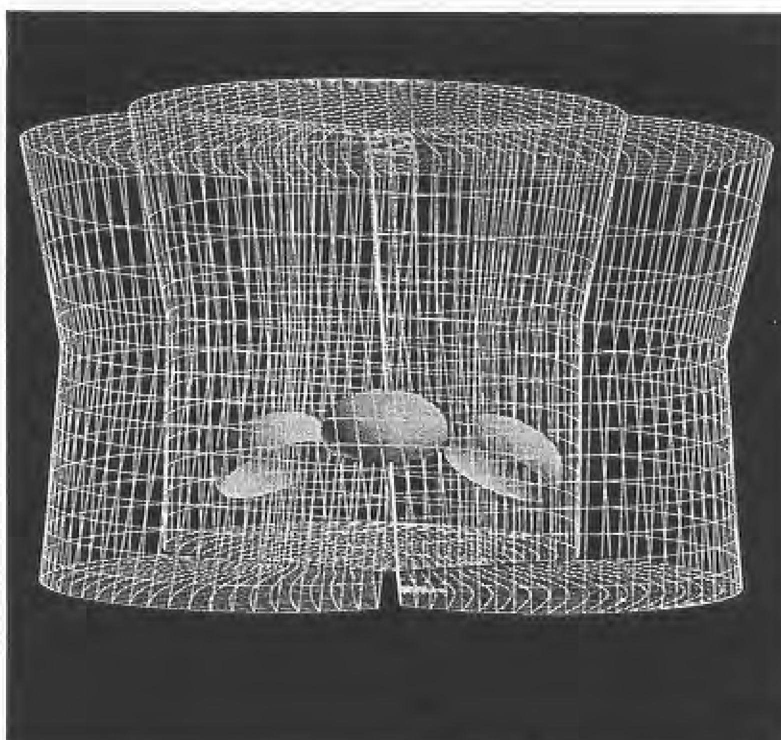

Figure 2. Computational Grid for a Parachute Cluster.

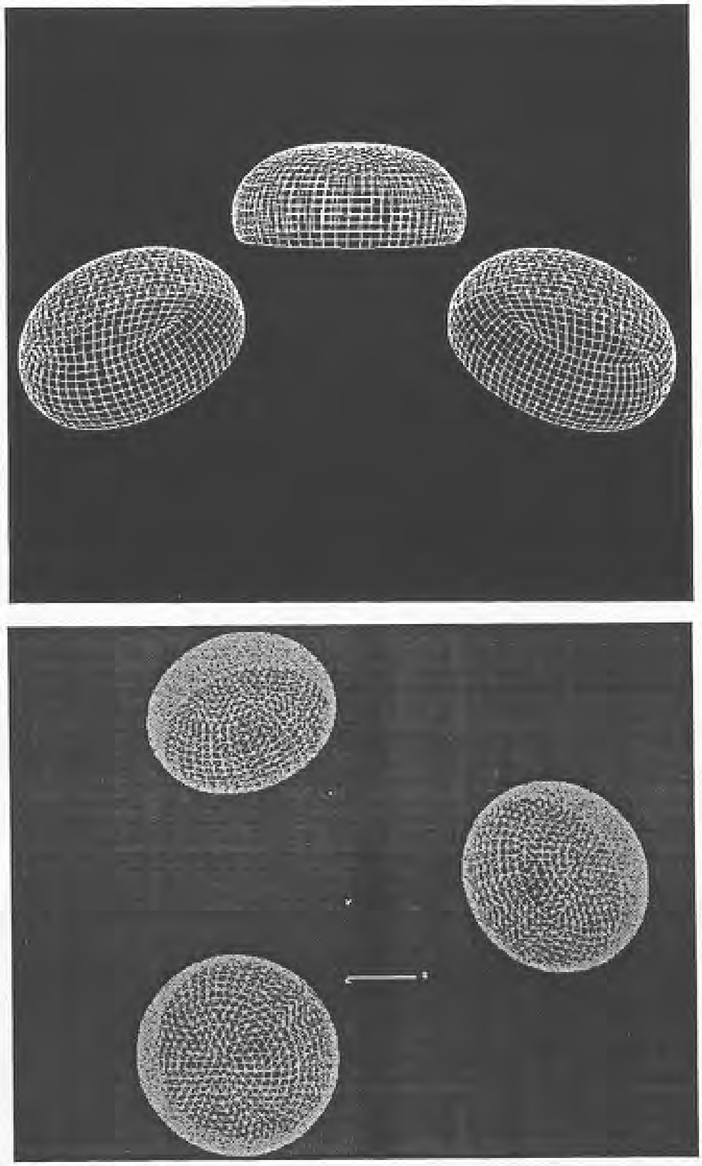

Figure 3. Surface Grids (structured and unstructured).

4. RESULTS

The input velocity used is 18 feet per second, which was the terminal descent velocity determined from the experimental results and is very close to the terminal descent velocity used in the axisymmetric models. All numerical computations were performed at this in-flow velocity and at $\alpha = 0^{\circ}$. Computed results have been obtained for the 3-D cluster configuration and are now presented. The computed results were obtained using the flow solvers described earlier. Although computations were performed on one parachute because of symmetry, the cluster consists of three canopies which are shown in Figure 3. Figure 4 shows the particle traces in the vicinity of the parachute. The incoming flow stagnates at the body surface, and a separated flow region is formed in front. It also shows flow separating at the skirt and then forming a large region of recirculatory flow behind the body. The flow in the wake is three-dimensional and shows the strong interactions in the wake region. This separated flow region in the wake gives rise to lower surface pressure on the outer body surface.

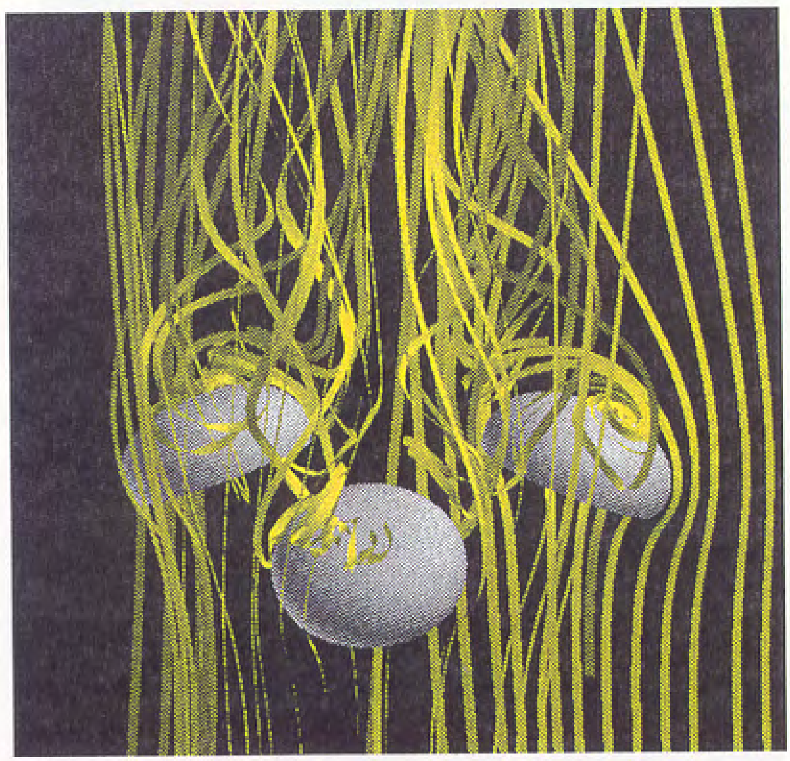

Figure 4. Particle Trace for Parachute Cluster, $\alpha = 0^{\circ}$.

The surface pressure distributions on the inner and outer surfaces of the canopies are shown in Figures 5 and 6, respectively. This case corresponds to a parachute polar angle of 25°, i.e., the canopies are 25° away from the vertical axis. As expected, the inner surface pressure (see Figure 5) is quite uniform and is a lot higher than the predicted pressure on the outer surface. The pressure on the outside of the canopies is low; also, there is a variation in the pressure from the apex to the skirt of the canopies (see Figure 6). The difference in pressure between the outer and the inner surfaces gives rise to drag force, which is consistent with the payload for this parachute at this terminal velocity. Currently, the canopy, suspension lines, and riser weights are incorporated into the payload weight.

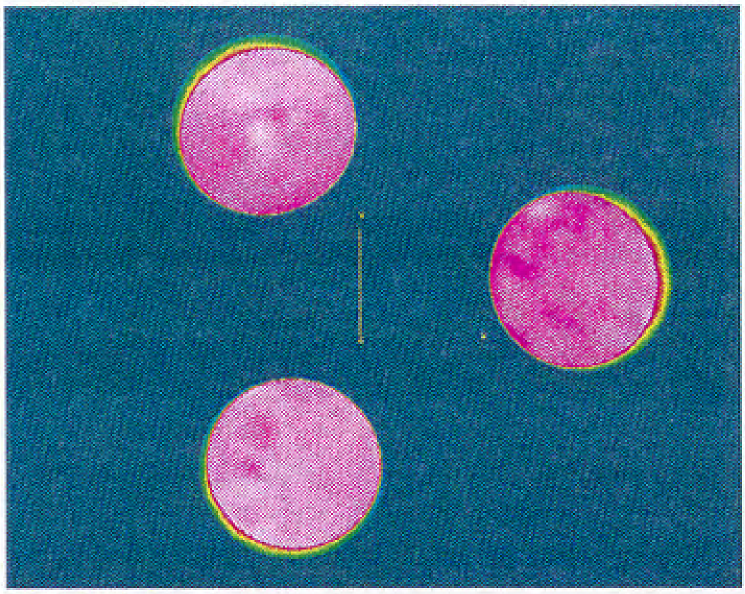

Figure 5. Computed Inner Surface Pressure, $\alpha = 0°$, ($\phi = 25°$).

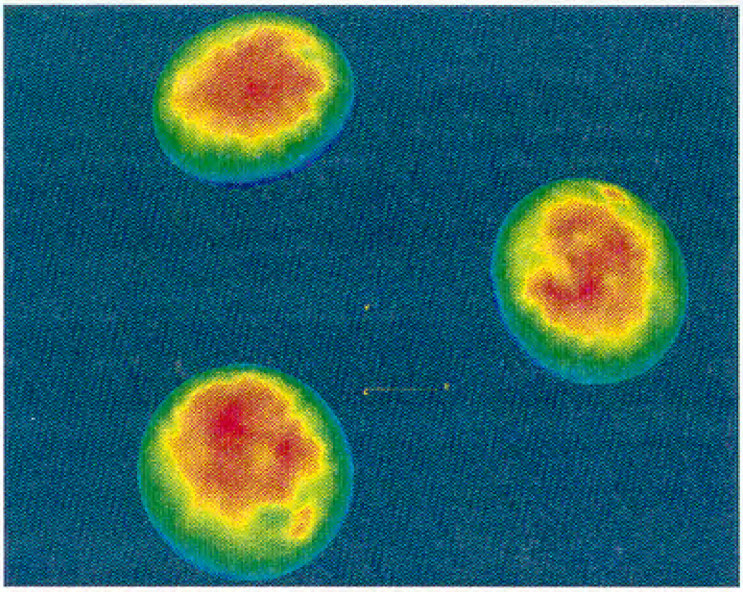

Figure 6. Computed Outer Surface Pressure, $\alpha = 0°$, ($\phi = 25°$).

Figure 7 shows the pressure contours along two planes cutting through the canopies. For the incoming flow, the pressure is uniform (free stream pressure). As the flow approaches the inner surface, it stagnates and forms a large region of high pressure in the vicinity of the inner surface. The pressure in the wake region is a low-pressure region. Although there is some variation of pressure in the low-pressure region, the outer surface pressure is a lot lower than the inner surface pressure. Sharp changes in the pressure field can be observed in going from the inner to the outer surface because of flow expansion.

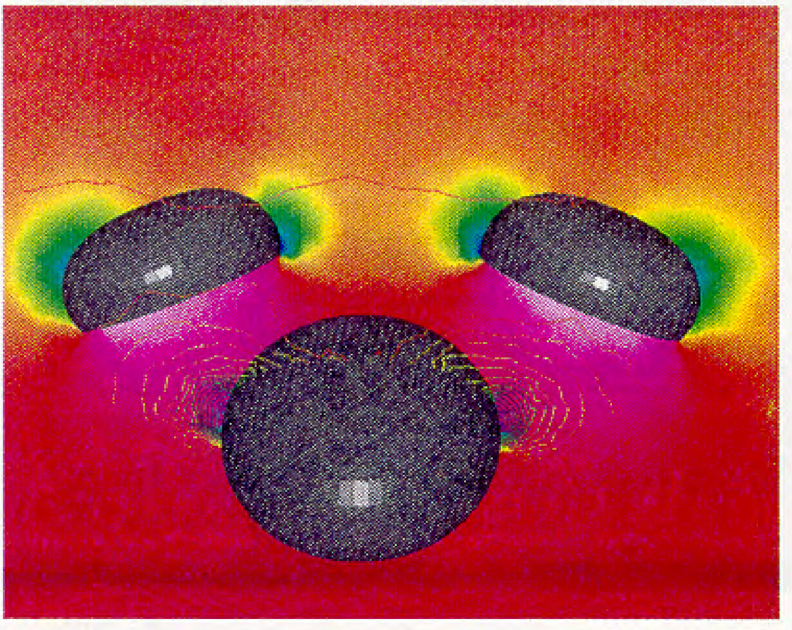

Figure 7. Computed Pressure Contours, $\alpha = 0°$, ($\phi = 25°$).

The computed pressure map for the parachute cluster is shown in Figure 8 for the polar angle of 40°. Again, it shows high pressure on the inside surface, which is rather uniform except for the skirt region. The computed pressure on the outer surface is lower and is not as uniform. As expected, the flow field is different compared to the 25° case. The pressure differential between the inner and the outer surface again determines the drag force for the parachute cluster at the prescribed terminal descent velocity.

As described in an earlier section, the forces and moments for the parachutes in cluster are determined from the computed solutions. Figure 9 shows the cluster efficiency factor as a function of the polar angle. The cluster efficiency factor is defined as the ratio of the drag of parachutes in a cluster to the drag of a single parachute times the number of parachutes, i.e., C_D (cluster) / [C_D (single) * number of parachutes]. The cluster efficiency factor is usually less than 1.0 because of wake flow field interactions. Figure 9 clearly shows the computed cluster efficiency factor in the range of 0.82 to 0.92, which is similar to the experimental data in Knacke[3] for larger solid round flat circular parachutes. The cluster efficiency drops a little with the increase in the polar angle. Computed results for $\phi = 25^\circ$ show that both the structured and unstructured solutions produce almost the same cluster efficiency factor.

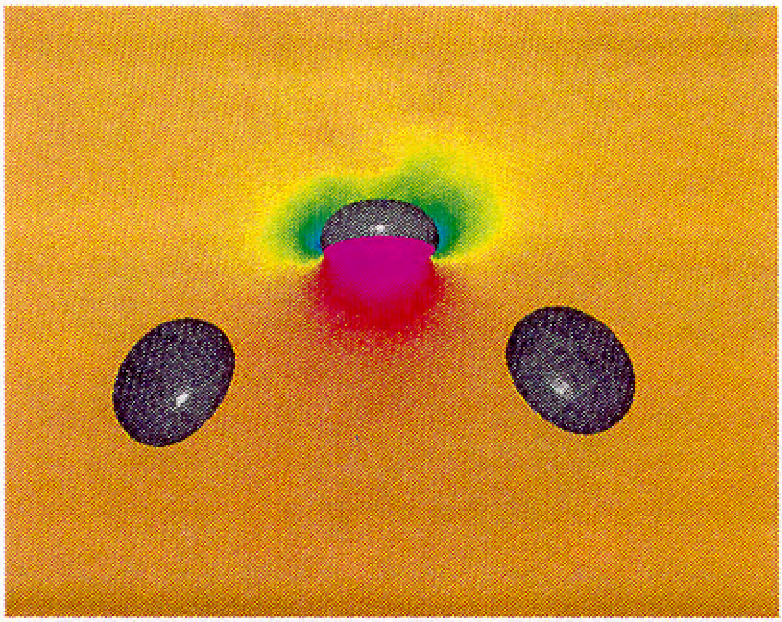

Figure 8. Computed Pressure Contours, $\alpha = 0°$, ($\phi = 40°$).

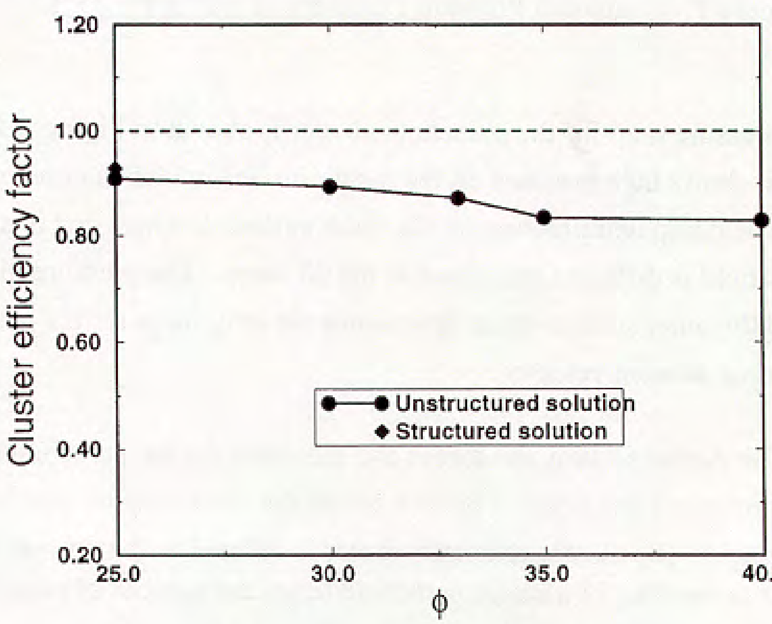

Figure 9. Cluster Efficiency Factor Versus Polar Angle, ϕ.

Figures 10 and 11 show the computed force and moment coefficients as a function of the polar angle of the clustered parachute configuration. These results are obtained from the unstructured flow computations for all three parachutes in a cluster. The three force components acting on the parachutes are shown in Figure 10. Here, the vertical force C_Y is the drag force and as expected, is the largest component. The predicted drag is almost constant with the polar angle. It is seen to decrease only a little with increased polar angle. The force components in the other two directions are rather small. It is interesting to note that both force components in the x and z directions are zero between polar angles of 30° and 35°. Figure 11 shows the variation of the moment components in three directions with the polar angle. The moment coefficients are referenced to the origin (0,0,0). All three moment components go through zero between polar angles of 30° and 35°. In particular, the component in y-direction is zero at $\phi = 33^\circ$. The expected stable configuration for the cluster geometry is determined by the condition at which the net moment about the payload (origin) is zero. In this case, it happens at approximately at $\phi = 32^\circ$. This computed polar angle compares very well with $\phi = 33^\circ$, which is the experimentally[8] observed polar angle for the stable configuration. In the experiments, the positions of the three canopies were measured during the opening process, and measured polar angle increased during the opening, until it stopped at around $\phi = 33^\circ$ when steady state was achieved.

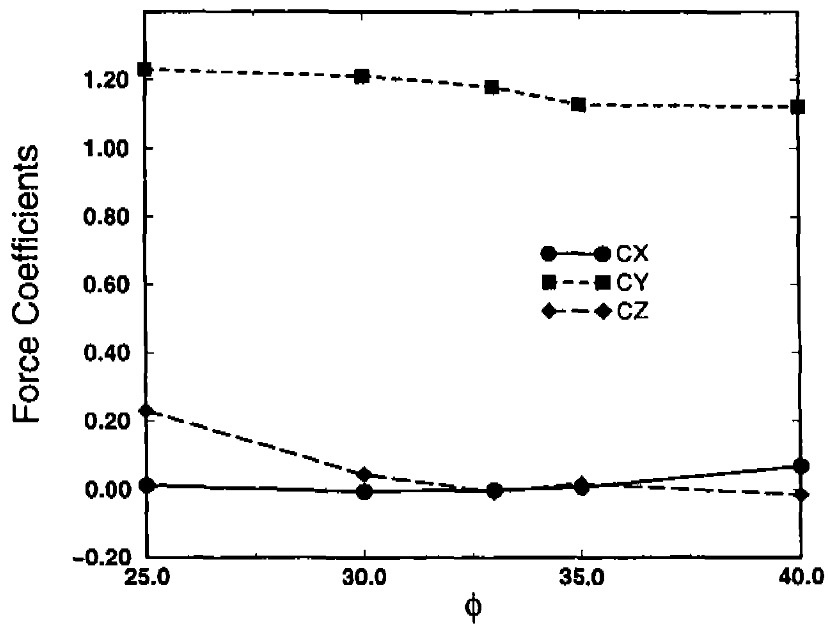

Figure 10. Force Components Versus Polar Angle, ϕ.

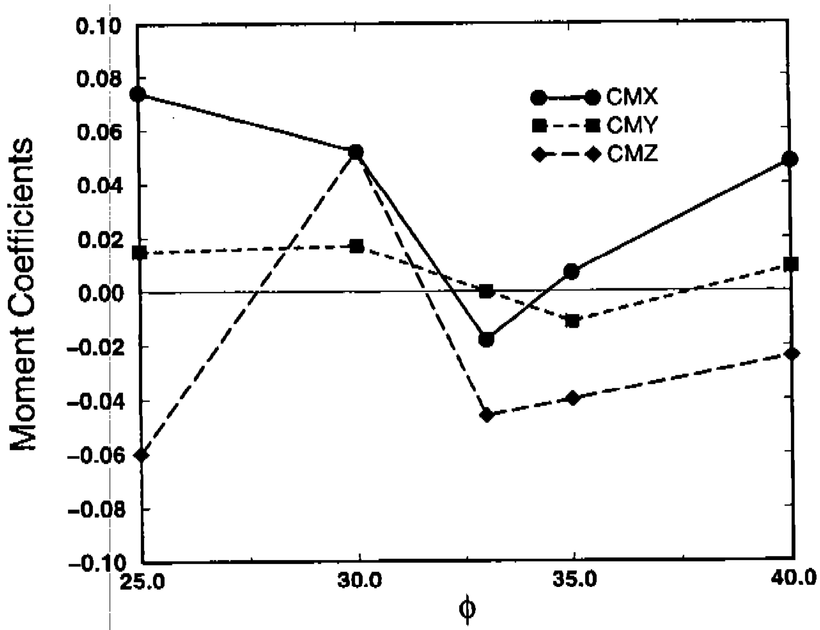

Figure 11. Moment Components Versus Polar Angle, ϕ.

5. CONCLUDING REMARKS

A computational study has been performed to determine the aerodynamics of a cluster of round parachutes using CFD. The results given here are the first predictions of descent characteristics for a cluster of three half-scale C-9 parachutes. In particular, the results include the aerodynamic flow field and geometry of the parachute cluster with assumed shapes for the individual canopies. Using periodic symmetry of the cluster configuration, the pressure over the inner and outer surfaces of a single canopy in the cluster is used to calculate the net vertical and radial forces acting on the canopy. A manual iterative process is used to determine the expected stable configuration for the cluster geometry by determining the condition at which the net moment about the payload (origin) is zero. The corresponding angular location of the canopies predicted by the CFD computations is compared with the available experimental results and is found to agree well with the data. It is shown that significant progress has been made in determining the terminal descent flow field characteristics of a particular parachute cluster configuration. These computational solutions provide, for the first time, an understanding of the flow field in and around parachute clusters.

This report has presented the current status of the modeling effort to address the complexity of the parachute characteristics and the associated 3-D flow fields of clusters of round parachutes. Future research will include the use of Chimera overset grid approach to compute the time-dependent fluid-dynamic forces associated with "rigid" parachutes in a cluster. This technique will allow the determination of the steady state stable configuration for a given cluster design "dynamically" within one numerical simulation. The developed technique is expected to assist in the development of future U.S. Army airdrop systems and other round parachute systems. The capability of accurately predicting the behavior of parachute systems will significantly reduce the amount of testing currently required.

INTENTIONALLY LEFT BLANK.

6. REFERENCES

1. Maydew, R. C., Peterson, C. W., and Orlik-Ruckemann, K. J., "Design and Testing of High-Performance Parachutes," AGARDograph No. 319, November 1991.

2. Cockrell, D. J., "The Aerodynamics of Parachutes," AGARDograph No. 295, July 1987.

3. Knacke, T. W., "Parachute Recovery Systems Design Manual," NWCTP 6575, Naval Weapons Center, China Lake, California, June 1987.

4. Ewing, E. G., Bixby, H. W., and Knacke, T. W., "Recovery Systems Design Guide," AFFDL-TR-78-151, December 1978.

5. Performance of and Design Criteria for Deployable Aerodynamic Decelerators, USAF ASD-TR-61-597, December 1963.

6. AIAA 1st Aerodynamic Deceleration Systems Conference, Houston, Texas, September 1966.

7. Vickery, E. D., and Eldredge, M. L., "Development of a System Clustered 137-ft Diameter Parachutes to Recover 60,000 lb," Proceedings of the 9th AIAA Aerodynamic Decelerator and Balloons Technology Conference, Albuquerque, NM, October 1986..

8. Lee, C. K., Lanza, J., and Buckley, J., "Apparatus and Method for Measuring Angular Positions of Parachute Canopies," Journal of Aircraft, vol. 33, no. 6, November-December 1996, pp. 1197-1199.

9. Lee, C. K., Lanza, J., and Buckley, J., "Experimental Investigation of Clustered Parachutes Inflation," Proceedings of the 14th AIAA Aerodynamic Decelerator Systems Technology Conference, San Francisco, CA, June 1997.

10. Stein, K. R., and Benney, R. J., "Parachute Inflation: A Problem in Aeroelasticity," U.S. Army Natick Research, Development, and Engineering Center. Natick Technical Report No. NATICK/TR-94/015, August 1994.

11. Sahu, J., Cooper, G., and Benney, R., "3-D Parachute Descent Analysis using Coupled CFD and Structural Codes," Proceedings of the 13th AIAA Aerodynamic Decelerator Systems Technology Conference, Clearwater, FL, May 1995.

12. Benney, R. J., and Leonard, J., "A 3-D Finite Element Structural Parachute Model," AIAA Paper No. 95-1563, 13th AIAA Aerodynamic Decelerator Conference, May 1995.

13. Pan, D., and Chakravarthy, S. R., "Unified Formulation for Incompressible Flows." AIAA Paper No. 89-0122, 27th Aerospace Sciences Meeting, Reno, NV, January 1989.

14. Szema, K. Y., Ramakrishnan, S. V., Shankar, V. V., and Rajagopal, K., "Application of a Generalized Lax-Wendroff Scheme for Unstructured Euler Computations," Proceedings of the AIAA 14th Applied Aerodynamics Conference, New Orleans, LA, June 1996.

NO. OF COPIES	ORGANIZATION
2	ADMINISTRATOR DEFENSE TECHNICAL INFO CENTER ATTN DTIC DDA 8725 JOHN J KINGMAN RD STE 0944 FT BELVOIR VA 22060-6218
1	DIRECTOR US ARMY RESEARCH LABORATORY ATTN AMSRL CS AL TA RECORDS MANAGEMENT 2800 POWDER MILL RD ADELPHI MD 20783-1197
1	DIRECTOR US ARMY RESEARCH LABORATORY ATTN AMSRL CI LL TECHNICAL LIBRARY 2800 POWDER MILL RD ADELPHI MD 207830-1197
1	DIRECTOR US ARMY RESEARCH LABORATORY ATTN AMSRL CS AL TP TECH PUBLISHING BRANCH 2800 POWDER MILL RD ADELPHI MD 20783-1197
2	USAF WRIGHT AERONAUTICAL LABORATORIES ATTN AFWAL FIMG DR J SHANG MR N E SCAGGS WPAFB OH 45433-6553
1	COMMANDER NAVAL SURFACE WARFARE CNTR ATTN CODE B40 DR W YANTA DAHLGREN VA 22448-5100
1	COMMANDER NAVAL SURFACE WARFARE CNTR ATTN CODE 420 DR A WARDLAW INDIAN HEAD MD 20640-5035
4	DIRECTOR NASA LANGLEY RESEARCH CENTER ATTN TECH LIBRARY MR D M BUSHNELL DR M J HEMSCH DR J SOUTH LANGLEY STATION HAMPTON VA 23665

NO. OF COPIES	ORGANIZATION
2	ARPA ATTN DR P KEMMEY DR JAMES RICHARDSON 3701 NORTH FAIRFAX DR ARLINGTON VA 22203-1714
7	DIRECTOR NASA AMES RESEARCH CENTER ATTN MS 227 8 L SCHIFF MS 258 1 T HOLST MS 258 1 D CHAUSSEE MS 258 1 M RAI MS 258 1 P KUTLER MS 258 1 P BUNING MS 258 1 B MEAKIN MOFFETT FIELD CA 94035
2	USMA DEPT OF MECHANICS ATTN LTC ANDREW L DULL M COSTELLO WEST POINT NY 10996
7	COMMANDER US ARMY ARDEC ATTN MCAR AET A R DEKLEINE C NG R BOTTICELLI H HUDGINS J GRAU S KAHN W KOENIG PICATINNY ARSENAL NJ 07806-5001
1	COMMANDER US ARMY ARDEC ATTN SMCAR CCH V PAUL VALENTI PICATINNY ARSENAL NJ 07806-5001
1	COMMANDER US ARMY ARDEC ATTN SFAE FAS SD MIKE DEVINE PICATINNY ARSENAL NJ 07806-5001
1	COMMANDER US NAVAL SURFACE WEAPONS CTR ATTN DR F MOORE DAHLGREN VA 22448
2	UNIV OF CALIFORNIA DAVIS DEPT OF MECHANICAL ENGG ATTN PROF H A DWYER PROF M HAFEZ DAVIS CA 95616

NO. OF COPIES	ORGANIZATION
1	AEROJET ELECTRONICS PLANT ATTN DANIEL W PILLASCH B170 DEPT 5311 P O BOX 296 1100 WEST HOLLYVALE STREET AZUSA CA 91702
3	SCIENCE AND TECHNOLOGY INC 4001 NORTH FAIRFAX DR NO 700 ATTN DR ALAN GLASSER MR BRUCE LOHMAN MR DAVE MAURIZI ARLINGTON VA 22203-1618
3	AIR FORCE ARMAMENT LAB ATTN AFATL/FXA STEPHEN C KORN BRUCE SIMPSON DAVE BELK EGLIN AFB FL 32542-5434
1	MASSACHUSETTS INSTITUTE OF TECHNOLOGY ATTN TECH LIBRARY 77 MASSACHUSETTS AVE CAMBRIDGE MA 02139
1	GRUMANN AEROSPACE CORP AEROPHYSICS RESEARCH DEPT ATTN DR R E MELNIK BETHPAGE NY 11714
2	MICRO CRAFT INC ATTN DR JOHN BENEK NORMAN SUHS 207 BIG SPRINGS AVE TULLAHOMA TN 37388-0370
1	LOS ALAMOS NATIONAL LAB ATTN MR BILL HOGAN MS G770 LOS ALAMOS NM 87545
3	DIRECTOR SANDIA NATIONAL LABORATORIES ATTN DIV 1554 DR W OBERKAMPF DIV 1554 DR F BLOTTNER DIV 1636 DR W WOLFE ALBUQUERQUE NM 87185

NO. OF COPIES	ORGANIZATION
1	NAVAL AIR WARFARE CENTER ATTN DAVID FINDLAY MS 3 BLDG 2187 PATUXENT RIVER MD 20670
1	METACOMP TECHNOLOGIES INC ATTN S R CHAKRAVARTHY 650 S WESTLAKE BLVD SUITE 200 WESTLAKE VILLAGE CA 91362-3804
2	ROCKWELL SCIENCE CENTER ATTN S V RAMAKRISHNAN V V SHANKAR 1049 CAMINO DOS RIOS THOUSAND OAKS CA 91360
1	ADVANCED TECHNOLOGY CTR ARVIN/CALSPAN AERODYNAMICS RESEARCH DEPT ATTN DR M S HOLDEN PO BOX 400 BUFFALO NY 14225
1	PENNSYLVANIA STATE UNIV DEPT OF AEROSPACE ENGG ATTN DR G S DULIKRAVICH UNIVERSITY PARK PA 16802
1	UNIV OF ILLINOIS AT URBANA CHAMPAIGN DEPT OF MECHANICAL AND INDUSTRIAL ENGINEERING ATTN DR J C DUTTON URBANA IL 61801
1	UNIVERSITY OF MARYLAND DEPT OF AEROSPACE ENGG ATTN DR J D ANDERSON JR COLLEGE PARK MD 20742
1	UNIVERSITY OF NOTRE DAME DEPT OF AERONAUTICAL AND MECHANICAL ENGINEERING ATTN PROF T J MUELLER NOTRE DAME IN 46556

NO. OF COPIES	ORGANIZATION
1	UNIVERSITY OF TEXAS DEPT OF AEROSPACE ENG MECH ATTN DR D S DOLLING AUSTIN TX 78712-1055
1	UNIVERSITY OF DELAWARE DEPT OF MECH ENGINEERING ATTN DR JOHN MEAKIN NEWARK DE 19716
1	UNIVERSITY OF FLORIDA DEPT OF ENGG SCIENCES COLLEGE OF ENGINEERING ATTN PROF C C HSU GAINESVILLE FL 32611
3	COMMANDER US ARMY SOLDIER SYSTEMS CMD NRDEC ATTN SSCM UTS R BENNEY K STEIN C LEE NATICK MA 01760-5017
	ABERDEEN PROVING GROUND
2	DIRECTOR US ARMY RESEARCH LABORATORY ATTN AMSRL CI LP (TECH LIB) BLDG 305 APG AA
27	DIR USARL ATTN AMSRL WM P A HORST E SCHMIDT AMSRL WM PB P PLOSTINS D LYON M BUNDY G COOPER E FERRY B GUIDOS K HEAVEY H EDGE V OSKAY A MIKHAIL J SAHU P WEINACHT AMSRL ST J ROCCHIO AMSRL WM PD B BURNS L BURTON AMSRL WM PA G KELLER M NUSCA AMSRL WM PC B FORCH AMSRL WM W C MURPHY AMSRL WM WB W D'AMICO AMSRL CI H C NIETUBICZ AMSRL CI HC P COLLINS D HISLEY D PRESSEL W STUREK

NO. OF COPIES	ORGANIZATION
2	CDR ARDEC ATTN FIRING TABLES R LIESKE R EITMILLER BLDG 120

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	November 1997	Final

4. TITLE AND SUBTITLE

Computational Fluid Dynamics Modeling of Parachute Clusters

5. FUNDING NUMBERS

PR: 1L161102AH43

6. AUTHOR(S)

Sahu, J. (ARL); Benney, R.J. (U.S. Army Soldier Systems Command); Ramakrishnan, S.V. (Rockwell International Science Center)

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

U.S. Army Research Laboratory
Weapons & Materials Research Directorate
Aberdeen Proving Ground, MD 21010-5066

8. PERFORMING ORGANIZATION REPORT NUMBER

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

U.S. Army Research Laboratory
Weapons & Materials Research Directorate
Aberdeen Proving Ground, MD 21010-5066

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

ARL-TR-1440

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

A computational tool that models the terminal descent characteristics of a single or a cluster of parachutes is a technology that is needed by parachute designers and engineers. As part of a technology program annex (TPA), a joint effort between the U.S. Army Natick Research, Development, and Engineering Center (NRDEC) and the U.S. Army Research Laboratory (ARL) to develop this computational tool is now under way. As a first effort, attempts are being made to analyze both two-dimensional (2-D) and three-dimensional (3-D) flow fields around a parachute using a coupling procedure in which the fluid dynamics are coupled to 2-D and 3-D structural dynamic (SD) codes. This effort uses computational fluid dynamic (CFD) codes to calculate a pressure field, which is then used as an input load for the SD code. Specifically, this report presents the methods and results of the flow field plus the structural characteristics of a single axisymmetric parachute and a 3-D gore configuration for the terminal descent velocity. Computed results have been obtained using the payload weight and unstretched constructed geometry of the canopies as input. Significant progress has been made in determining the terminal descent flow field along with the terminal shape of the parachute. A discussion of the fluid and structural dynamics codes, coupling procedure, and the associated technical difficulties is presented. Examples of the codes' current capabilities are shown.

14. SUBJECT TERMS

computational fluid dynamics — parachute clusters
low speed flow — terminal descent numerical computations

15. NUMBER OF PAGES

33

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. Z39-18
298-102