

NASA Contractor Report 191529
ICASE Report No. 93-62

2
AD-A272 674



ICASE



CLOUDING TRACING: VISUALIZATION OF THE MIXING OF FLUID ELEMENTS IN CONVECTION-DIFFUSION SYSTEMS

Kwan-Liu Ma
Philip J. Smith

S DTIC
ELECTE
NOV 16 1993
A

This document has been approved
for public release and sale; its
distribution is unlimited.

NASA Contract No. NAS1-19480
August 1993

Institute for Computer Applications in Science and Engineering
NASA Langley Research Center
Hampton, Virginia 23681-0001

Operated by the Universities Space Research Association



National Aeronautics and
Space Administration
Langley Research Center
Hampton, Virginia 23681-0001

*Original contains color
plates. All DTIC reproductions
will be in black and
white.*

93-28044



93 11 15 073

Clouding Tracing: Visualization of the Mixing of Fluid Elements in Convection-Diffusion Systems

Kwan-Liu Ma[†]

Institute for Computer Application in Science and Engineering

NASA Langley Research Center

Philip J. Smith

Department of Chemical and Fuels Engineering

University of Utah

DTIC QUALITY INSPECTED 5

DTIC QUALITY INSPECTED 6

Accession For	
NTIS CRA&I	<input checked="" type="checkbox"/>
DTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By	
Distribution/	
Availability Codes	
Dist	Avail and/or Special
A-1	

Abstract

This paper describes a highly interactive method for computer visualization of the basic physical process of dispersion and mixing of fluid elements in convection-diffusion systems. It is based on transforming the vector field from a traditionally Eulerian reference frame into a Lagrangian reference frame. Fluid elements are traced through the vector field for the mean path as well as the statistical dispersion of the fluid elements about the mean position by using added scalar information about the root mean square value of the vector field and its Lagrangian time scale. In this way, clouds of fluid elements are traced not just mean paths. We have used this method to visualize the simulation of an industrial incinerator to help identify mechanisms for poor mixing.

[†]This research was supported in part by the National Aeronautics and Space Administration under NASA contract NAS1-19480 while the author was in residence at the Institute for Computer Application in Science and Engineering (ICASE), NASA Langley Research Center, Hampton, VA 23681-0001.

1 Introduction

Many computational simulations in engineering involve solutions of simultaneous vector and scalar fields for convection and diffusion equations in three dimensional space. The physical process of dispersion and mixing within such systems is often what the engineer is trying to understand when performing the numerical simulation. Current computer visualizations of calculated scalar and vector fields do not provide sufficient insight to immediately conceptualize the underlying dispersion and mixing process inherent in these systems. For example, in computational fluid dynamics problems involving the mixing and reaction of three input streams (A, B and C), the user would like to know how quickly stream A disperses, where it is going, and if and where it mixes with streams B and C which are also convecting and dispersing. Schemes to visualize the scalar concentration of stream A tell something about the extent of mixing but nothing about where A came from or with which stream (B or C) it has mixed. Particle tracing methods can give an idea of where the mean flow path is found for a representative massless fluid element starting at any point (i.e. in stream A) but nothing about its dispersion rate or its interaction (mixing) with other streams.

In this paper a highly interactive visualization scheme is presented which attempts to allow a computational fluid dynamics user to visualize the basic physical process of dispersion and mixing rather than just the vector or scalar values computed by the simulation. The motivation for this approach is the hope that if the user can gain a greater insight into convection and diffusion through the graphical visualization, then the original simulation will be of greater utility in solving practical engineering problems. As shown in our test results, this visualization method allows us to identify faults in the design of of an industrial incinerator.

The method is based on transforming the vector field from a traditionally Eulerian (fixed) reference frame into a Lagrangian (moving) reference frame. Mean fluid elements are traced through the computed vector field (as in traditional particle tracing methods). The statistical dispersion of the fluid elements about the mean position is also computed by using added scalar information about the root mean square value for the vector field and its Lagrangian time scale. These scalar values can usually be computed easily from existing information in computational fluid dynamics codes (i.e. turbulence models, etc.). In this way clouds or plumes of fluid elements are traced not just mean paths.

The cloud surface is represented as a triangular mesh which can be efficiently rendered by conventional graphics hardware. The surface is colored by using any scalar field such as temperature, pressure, etc. selected by the user. During a visualization session, the user can interactively pick multiple three-

dimensional positions in the data domain and see how flows starting from these positions develop and mix together in some way. Local twisting or deformation can be visualized by displaying ribbons along the mean trajectory. The interactive cloud tracing scheme presented in this paper complements existing visualization techniques.

2 Flow Visualization

As both the cost of experimental flow visualization and the scale of the problems that scientists can simulate continue to increase, flow visualization using computer graphics techniques becomes an essential and popular mechanism for scientific researchers to understand the results of fluid dynamics simulations in fields such as aeronautics, transportation design, weather forecasting, oceanography, combustion engineering, etc. Well known flow visualization methods making use of computer graphics include arrow fields, contouring, streamlines, particle tracing, etc. To date, many elaborate methods for graphical flow visualization have been developed, but none of them has emerged as being the standard due to the complexity of three-dimensional flow fields.

Stream surfaces/ribbons [9] provide information about twisting along particle paths. The stream polygon method [14] displays local deformation due to normal and shear strain and rigid body rotation. Silver et al. [15, 18] describes how to visualize vortex structures by employing so called visiometric techniques including thresholding, object isolation, abstraction and data juxtaposition.

The concept of using many moving particles to create the illusion of fluid movement originated from Reeves [13] Spot noise [17] is the use of stochastic texture (overlapping particles) for the visualization of scalar and vector fields over surfaces. A spot is a two-dimensional pattern like a disk, a square or pair of crossed lines. The attributes of spots such as size, shape, direction, and edge are varied according to local scalar and vector flow fields resulting in directional information, which provides a qualitative visualization of the data. Particle-based rendering is usually computationally expensive. Crawfis and Max [7, 11] develop an efficient technique integrating a fine-grained texture representation of vector flow field into a direct volume rendering method to visualize vector and scalar fields simultaneously. Gelder and Wilhelms [16] present a highly interactive flow animation technique also based on the concept of particle systems. Particles are placed randomly in the flow field. Each particle path is represented as a polyline. Then particle motion is created by using a color-table animation technique.

Critical points are locations in the flow field where the velocity field vanishes. These points can be characterized according to the nearby integral curves. Critical points in a flow field along with the

integral curves connecting them may completely describe the topology of the vector field. Much work has been done in the visualization of two-dimensional flow topology and three-dimensional separation flows near the surface of a body [4, 6, 8]. The integration of critical-point analysis into a graphical flow visualization process also provides a powerful divide-and-conquer approach to data exploration since the topological information can be used to partition a large and complex flow field into well-defined regions.

Virtual smoke is an interactive flow tracing technique that we developed previously [10]. It simulates the use of colored smoke for experimental gaseous fluid flow visualization. It is based on direct volume rendering so realistic three-dimensional visualization of the flow can be produced. In particular, it can animate the dynamic behavior of steady-state or instantaneous flow fields. While the virtual smoke method can display realistic flow patterns in motion, its diffusive component is less accurate and it in fact can produce diffusive flow artificially.

The cloud tracing method presented in this paper is a revision of virtual smoke to make visualizations that are more physically accurate. The next section gives a brief description of the physical model and the related mathematical equations behind the cloud tracing visualization method. Then the graphics algorithm used to realize the cloud tracing visualization is described in Section 4. We have applied this tracing technique to a data set from the simulation of an industrial incinerator and the results are reported in Section 5.

3 A Dispersion Model

The dispersion of particles or fluid elements by turbulent motion has been an active field of research in recent years. Practical interests are widespread and include the combustion of particulate and droplet fuels, the pneumatic transport of particles, the mixing and settling of industry emissions, particle deposition, electrostatic precipitation, and particle fluid separations. Practical combustion processes like spray combustion, pulverized coal and coal slurry combustion, fluidized beds, sorbent injection, and hazardous waste incineration, introduce the fuel as particles, liquid droplets, or slurries into turbulent environments.

Mixing and dispersion of gaseous streams involves fluid elements that disperse like particles but without any added inertial effects of particles with drag. The equations used here were developed by Batchelor [1]. They have been extended and used for turbulent dispersion of inertial particles in a model for the stochastic transport of particles, referred to as the STP model [2]. This approach is a stochastic Lagrangian scheme that can accurately and efficiently compute fluid dispersion in complex turbulent

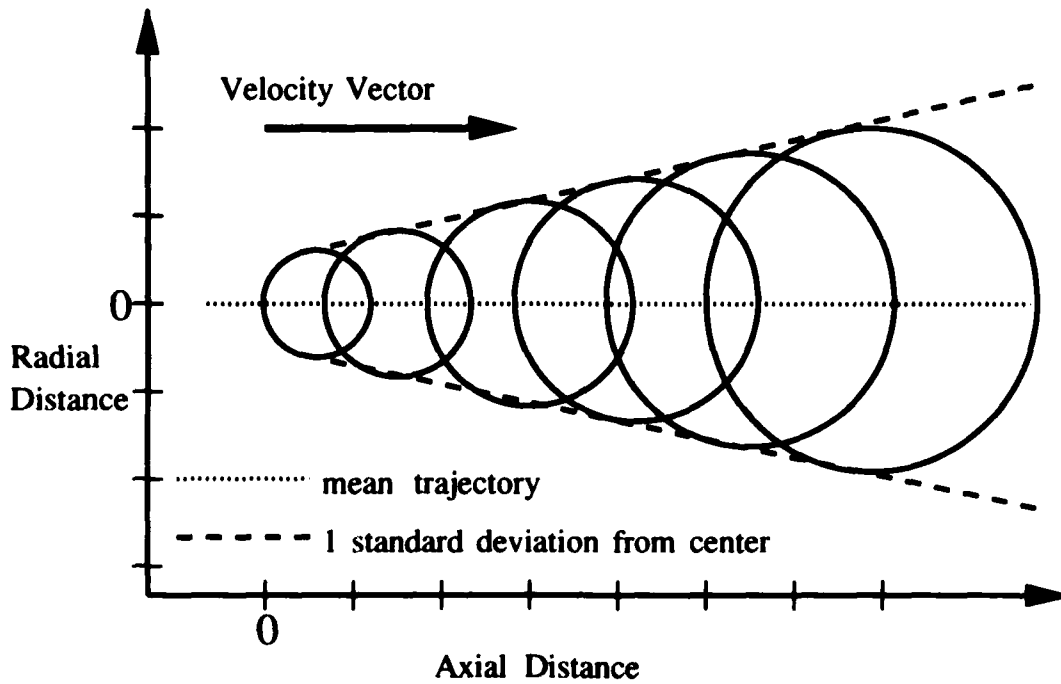


Figure 1: A Lagrangian View of Dispersion in a One-dimensional Flow

flows.

In this approach, the time evolution of a probability density function for fluid particle position is modeled. Figure 1 illustrates pictorially the development of a probability density function for particle position for an initially concentrated ensemble of fluid particles released from a source near the origin in a two dimensional coordinate system. The circles circumscribe the region of the flowfield included within one standard deviation of the mean fluid particle position, which is at the center of each circle. A brief mathematical description of the approach is given in the following section.

The path of an individual fluid element is given by

$$x(t) = \int_0^t V(\tau) d\tau + x(0) \quad (1)$$

where V is fluid velocity and x is particle position.

Then, for a stationary velocity field, the variance of the probability density function for fluid particle position can be written as

$$\sigma(t)^2 = 2 \int_0^t (t - \tau) R(\tau) d\tau \quad (2)$$

$R(\tau)$ is the fluid particle autocorrelation function and takes the form

$$R(\tau) = v_{rms}^2 e^{-\tau/T_L} \quad (3)$$

where v_{rms} is the root mean square fluid velocity fluctuation, and T_L is the Lagrangian integral time scale which is the average persistence of the turbulence activity at a point. The root mean square gas velocity fluctuations are assumed to be isotropic and are calculated from the turbulent kinetic energy, k , as

$$v_{rms}^2 = \frac{2k}{3}$$

T_L comes from the description of the turbulent field. For example, if a k - ϵ model is used,

$$T_L = \frac{C_\mu^{3/4} k^{3/2}}{\epsilon (\frac{2}{3}k)^{1/2}}$$

where C_μ is a constant and ϵ is the computed dissipation rate of turbulence.

4 Visualization of Cloud Tracing

Unlike traditional computer graphics, the goal of visualization is *insight*, not realistically rendered images. The focus of the following discussion is on a surface rendering technique, which allows the user to depict how fluid elements retain certain properties in a highly interactive manner. The surface normally appears as a growing cone which is deformed due to fluctuating local vector fields. By using surface rendering, we sacrifice realism and hope that higher interactivity and flexibility can better assist the user in obtaining insight into the flow fields. As described later, more realistic visualization of the flow can still be obtained by using a direct volume rendering technique similar to virtual smoke. By any means, the fluid particle dispersion is calculated statistically and the diffusive behavior of the flow is described more accurately.

4.1 Tracing

The cloud tracing starts from a three-dimensional position $P(x, y, z)$ specified by the user using a three-dimensional cross hair. At this position, V , v_{rms} and T_L are trilinearly interpolated from the neighboring eight voxels. As in traditional fluid particle tracing, knowing V , P and a user-specified time increment dt , the mean particle trajectory T can be calculated using a numerical integration method like the two-step Runge-Kutta time-advance method such that:

$$\begin{aligned} P^* &= P_i + dtV(P_i) \\ P_{i+1} &= P^* + \frac{1}{2}dt(V(P^*) - V(P_i)) \end{aligned}$$

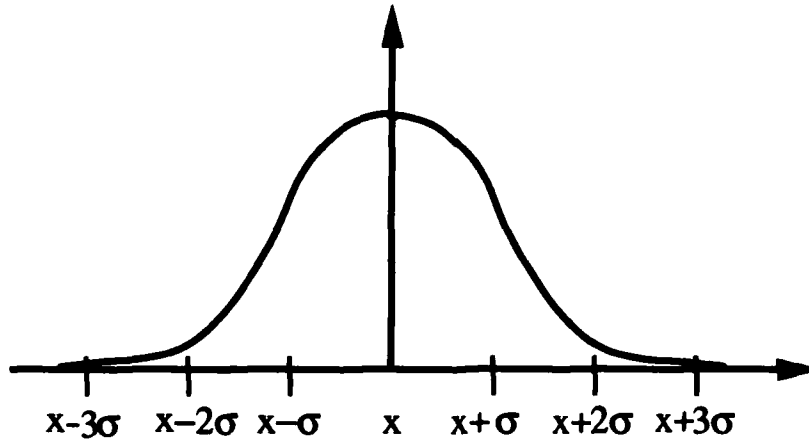


Figure 2: Dispersion Area as a Function of Variance within 3 Standard deviation.

Thus T contains a sequence of points $P_i(x, y, z)$ along the trajectory calculated where $P_0 = P$. At each mean point P_i , we then calculate the dispersion of the fluid elements about P_i by using scalar quantities v_{rms} and T_L . Equation 2 is used for this calculation. $\sigma(t)$ defines the spatial extent of the probability density function for particle position at successive and equally spaced residence times. Intuitively, $\sigma(t)$ defines a circle circumscribing the region of the flow field included within one standard deviation of the mean particle position. That is, P_i is the center and $\sigma(t)$ is the radius of the circle. Figure 2 shows the dispersed area as a *bivariate normal* distribution function of the variance within three standard deviation of the mean particle position. The integration is done by polynomial interpolation. Using the Trapezoid method, Equation 2 becomes

$$\begin{aligned}\sigma(t)^2 &= 2 \int_0^t (t - \tau) R(\tau) d\tau \\ &= \frac{h}{2} (tR(0) + 2 \sum_{i=1}^{n-1} (t - ih) R(ih))\end{aligned}$$

where $h = \frac{t}{n}$ and n is the number of points.

In actual implementation, the trajectory is calculated incrementally along with the calculation of the variance of the probability density function. For each pair of consecutive points P_i and P_{i+1} , a triangle mesh connects the circumference of the two circles defined at the corresponding points. The orientation of a circle is defined by the velocity at the corresponding point. Applying a triangle mesh along the trajectory in this fashion generates a cone-shape surface which represents the dispersion along the trajectory.

4.2 Surface Construction

The way the surface is constructed is similar to the traditional method for extracting surfaces between consecutive contours [12]. However, there is no ambiguity in our case since the consecutive slices to be connected are always perfect circular planes with no holes. Our algorithm uses a standard circular plane which is transformed from point to point according to the local velocity $V = (u, v, w)$. We define the standard circular plane on the xy plane and centered at the origin of the corresponding Cartesian coordinate system. In addition, the derivation of a transformation matrix for the rotation of a vector about a general axis through the origin that is described in Faux's book [5] can be used for our purpose. Because the orientation of the standard plane is $(0, 0, 1.0)$, the transformation matrix is greatly simplified as follows:

$$\begin{bmatrix} v^2(1-w) + w & -uv(1-w) & \sin(\theta)u & 0 \\ -uv(1-w) & u^2(1-w) + w & \sin(\theta)v & 0 \\ -\sin(\theta)u & -\sin(\theta)v & w & 0 \\ x & y & z & 1 \end{bmatrix}$$

where u, v , and w are normalized, $\theta = \sqrt{u^2 + v^2}$ which is the angle between $(0, 0, 1.0)$ and (u, v, w) , and x, y , and z are the coordinates of P_i . Figure 3 shows the standard plane and a sequence of transformations. Note that only the points $C_i(n)$ approximating the circle are transformed. The number of points used is selected by the user. Alternatively, the number of points can grow adaptively according to the radius of the circle; however, care must be taken to ensure continuity of the triangle mesh. Note that a 4-point model would resemble the stream polygon visualization [14] but is not sheared. The surface normal is approximated by calculating $C_i(n) - P_i$ at each $C_i(n)$. The surface is hardware Gouraud shaded. The interpolated scalar value at each triangle point is mapped to the color selected by the user. Figure 4 shows the wireframe representation of two cloud surfaces.

In addition, a ribbon tracer can be made by consistently selecting two diagonally opposite points on each circle and then constructing a surface using those points (spinwise edges) as shown in Figure 5. A cross ribbon tracer can be made by selecting two sets of such diagonal points on each circle. However, the two lines crossing the two pairs of points on each circle should be orthogonal. Besides being used to examine local twisting and deformation, ribbons may also help observe flow mixing. The cloud tracing technique described above has been implemented in Motif and GLX for running on Silicon Graphics workstations. Motif is used to implement the user interface. GLX which allows GL and X mixed-mode programming is used to implement 3D graphics. On a workstation supporting *alpha blending*, transparent surfaces can be used to derive a more intuitive visualization.

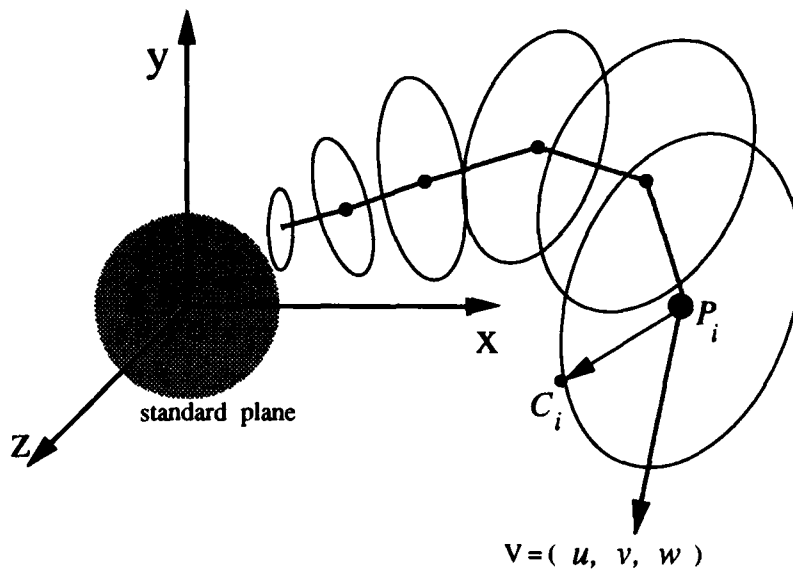


Figure 3: Standard Plane and Transformations.

A realistic volume rendered image of cloud tracing can still be obtained after the traces are made by using the technique described in [3, 10]. Essentially, the mean particle paths $P_i(t)$ are treated as seedlings and $\sigma(t)$ is used as the radius to the corresponding seed. The result is an improved virtual smoke which accurately visualizes the convection as well as diffusive flow. Nevertheless, interactive direct volume rendering is still limited to a fixed view position.

5 Test Results

A computer simulation of an organic waste incinerator was performed. In this cylindrical furnace the waste is injected into the combustion chamber through 28 burners located around the perimeter and near the base of the furnace. Each burner has two concentric annuli; the waste stream is injected through the center and air through the outer annulus. The objective of the incinerator design is to accomplish complete mixing and reaction of the waste and air streams as rapidly as possible. The objective of the visualization is to explore the success or failure of the design for accomplishing this rapid mixing. In this test, because of the symmetry of the furnace design and the area of interest, only one side of the lower half of the furnace is visualized. This portion of the furnace contains 100,800 data points in cylindrical coordinates.

Plate 1 shows traditional fluid particle traces originating in: 1) the center of the waste stream and 2) the center of the air stream, for one of the 28 burners. Although the fluid path is shown in this method, it

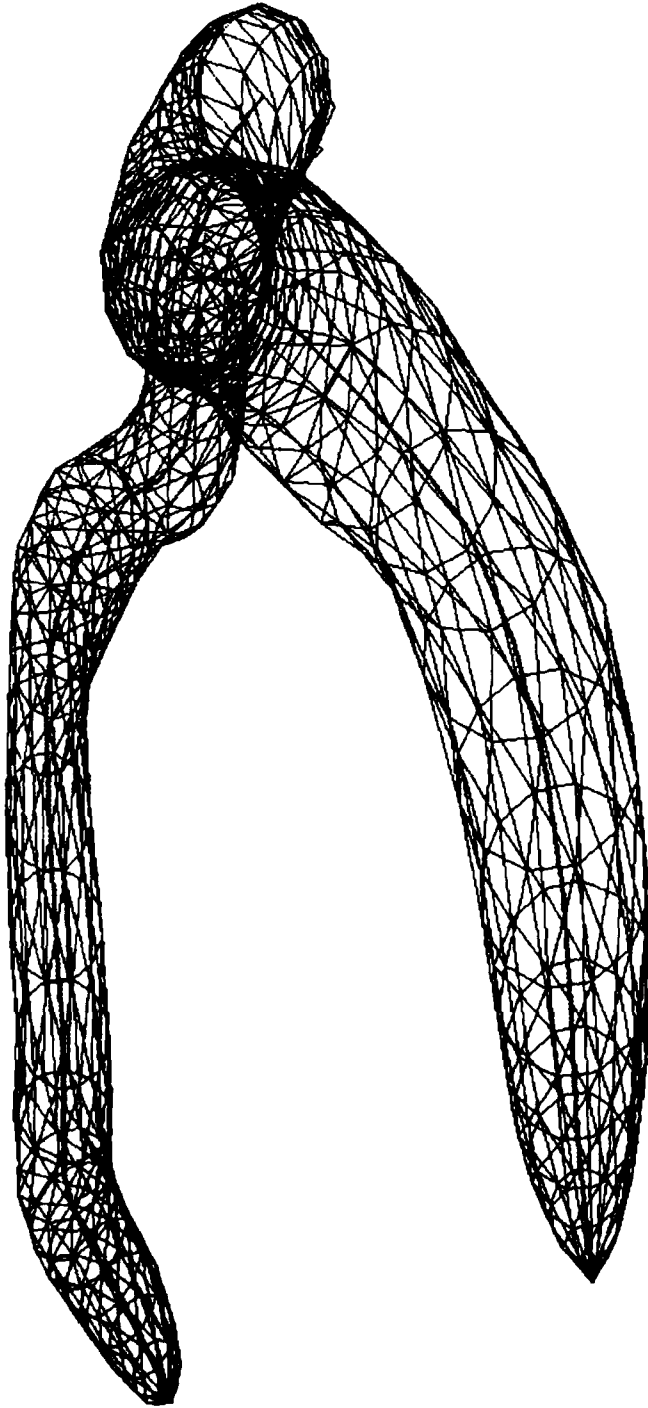


Figure 4: Cloud Construction.

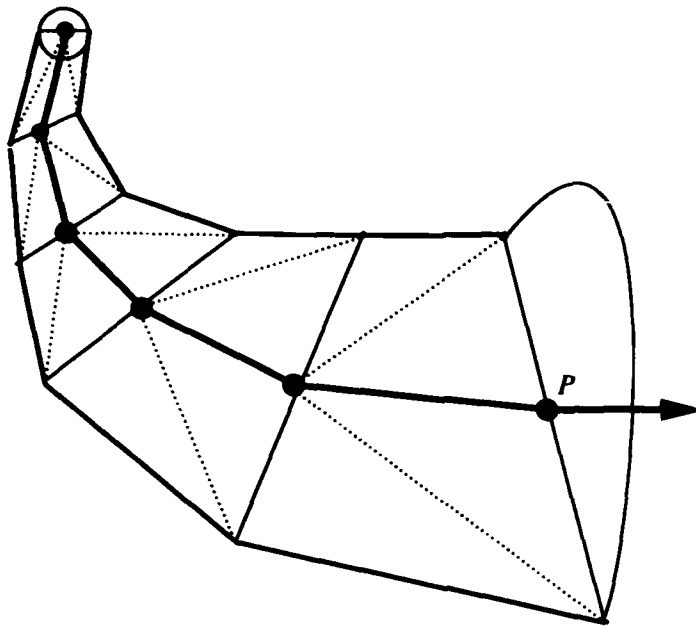


Figure 5: Ribbon Construction.

gives little information on the extent of mixing between the two streams. If we track more fluid elements we still do not gain any better insight into the path of the cloud since the paths that are traced only represent the mean location for any group of fluid elements released from the initial location. The actual mixing includes the statistical dispersion (convection and diffusion) of the entire cloud released from the specified location.

Plate 2 shows the cloud tracing performed by the technique discussed in this paper. The surface around each mean path encloses one standard deviation (68.27%) of each cloud. Now the extent of mixing of these two clouds is apparent. Plate 3 shows a view that is rotated to look from the top of the incinerator. Because this furnace was designed with so many burners and with both air and waste in each burner, it was expected that mixing would be accomplished rapidly. Both experimental testing and numerical simulation showed otherwise. The objective of the simulation was to see why mixing was not as rapid as was expected. Simple color shading of the waste concentration at the exit plane of the furnace showed high waste concentrations near the center of the furnace and high concentration of air at the outer perimeter of the furnace. The cloud tracing visualization identified the reasons for this. The two clouds shown in Plate 2 and 3 show that the waste stream from each burner penetrates further into the center of the cylindrical furnace before turning upward by the mean flow from below; while the air stream on the outside of each burner turns upward much sooner and stays at the outer perimeter of

the furnace. The cloud surface is shaded by mapping O_2 concentration to colors; low concentration to white-blue and high to red-purple.

The rendering of the dispersing cloud as a colored surface makes real time interactivity possible. Interactivity of the visualization scheme is necessary for the analysis of complex mixing chambers. The spatial relation of the dispersing stream tube can only be appreciated when the rendered image can be rotated in real time. The scalar field is mapped to color on the tube surface. Real time rotations allow a full exposure of the colored surface to the observers' view. The color of the surfaces in Plate 2 and 3 is representative of the air concentration. Thus the visualization indicates simultaneously that the two jets shown do not mix readily; but instead, air from other surrounding jets tends to stay at the outer radius of the incinerator. Plate 4 shows a surface enclosing 2.0 standard deviations (95%) for each of the clouds traced. Only the outer portions of these two representative clouds begin to mix by the end of this furnace.

This visualization scheme has been able to uncover mechanistic reasons for the poor mixing in an industrial incinerator. The poor mixing was not expected from an engineering analysis of the initial design. The reasons for the poor mixing were not uncovered from experiments performed with the incinerator or from the numerical simulation of the furnace without this visualization tool.

6 Conclusions

Interactivity is the key to successful scientific visualization. We have introduced a highly interactive cloud tracing technique for the visualization of simultaneous vector and scalar fields in convection-diffusion systems. A dispersion model for fluid elements is integrated into the visualization process to more accurately portray the physical process of dispersion and mixing within such system. The use of a surface rendering technique allows the user to move the cloud tracer in real time. By using a data set from the simulations of an industrial incinerator, we have demonstrated that this cloud tracing technique is an effective analysis tool for practical engineering problems. It gives visualization to the mechanism of gaseous mixing not just the result of mixing.

Future work includes a better user interface for locating seed points, the integration of volume rendering for viewing more than the cloud surfaces, tracing of particle cloud and enhancement for handling unsteady flows. For unsteady flows, clouds must evolve simultaneously and not sequentially as in our current implementation.

Acknowledgments

Thanks go to Sandeep Jain for his assistance in preparing the test data sets. Thanks also go to Elena Driskill, Jamie Painter, K. Sikorski and Bart Singer for valuable comments. This work has been supported in part by NSF/ACERC and NASA/ICASE.

References

- [1] BATCHELOR, G. Diffusion in Free Turbulent Shear Flows. *Journal of Fluid Mechanics* 67, 3 (1957).
- [2] BAXTER, L. *Turbulent Transport of Particles*. PhD thesis, Brigham Young University, Provo, Utah, 1992.
- [3] COHEN, M. F., PAINTER, J. S., MEHTA, M., AND MA, K.-L. Volume Seedlings. In *Proceedings of ACM Symposium on Interactive 3D Graphics* (March 1992).
- [4] DICKINSON, R. R. Interactive Analysis of the Topology of 4D Vector Fields. *IBM Journal of Research and Development* 35, 1 (January 1991), 59-66.
- [5] FAUX, D., AND PRATT, M. *Computational Geometry for Design and Manufacture*. Ellis Horwood, 1979.
- [6] GLOBUS, A., AND LEVIT, C. A Tool for Visualizing the topology of Three-dimensional Vector Fields. In *Proceedings Visualization'91* (1991), G. M. Nielson and L. Rosenblum, Eds., IEEE Computer Society Press, pp. 33-40.
- [7] GRAWFIS, R., AND N., M. Direct Volume Visualization of Three-Dimensional Vector Fields. In *1992 Workshop on Volume Visualization* (1992), pp. 55-60.
- [8] HELMAN, J. L., AND HESSELINK, L. Visualization of Vector Field Topology in Fluid Flows. *IEEE Computer Graphics and Applications* 11, 3 (May 1991), 36-46.
- [9] HULTQUIST, J. Constructing Stream Surfaces in Steady 3-D Vector Fields. In *Proceedings of Visualization'92* (1992), A. Kaufman and G. M. Nielson, Eds., pp. 171-178.
- [10] MA, K.-L., AND SMITH, P. J. Virtual Smoke: An Interactive 3D Flow Visualization Technique. In *Proceedings of Visualization'92* (1992), A. Kaufman and G. M. Nielson, Eds., pp. 46-53.

- [11] MAX, N., GRAWFIS, R., AND WILLIAMS, D. Visualizing Wind Velocities by Advecting Cloud Textures. In *Proceedings of Visualization'92* (1992), A. Kaufman and G. M. Nielson, Eds., pp. 179-184.
- [12] MEYERS, D., AND SKINNER, S. Surfaces from Contours. *ACM Transactions on Graphics* 11, 2 (July 1992), 228-258.
- [13] REEVES, W. Particle Systems - A Technique for Modeling a Class of Fuzzy Objects. In *Computer Graphics* (1983), vol. 17, pp. 359-376.
- [14] SCHROEDER, W., VOLPE, C., AND LORENSEN, W. The Stream Polygon: A Technique for 3D Vector Field Visualization. In *Proceedings Visualization'91* (1991), G. M. Nielson and L. Rosenblum, Eds., IEEE Computer Society Press, pp. 126-132.
- [15] SILVER, D., GAO, M., AND ZABUSKY, N. Visualizing Causal Effects in 4D Space-Time Vector Fields. In *Proceedings Visualization'91* (1991), G. M. Nielson and L. Rosenblum, Eds., IEEE Computer Society Press, pp. 12-16.
- [16] VAN GELDER, A., AND WILHELMS, J. Interactive Animated Visualization of Flow Fields. In *1992 Workshop on Volume Visualization* (1992), pp. 47-54.
- [17] VAN WIJK, J. J. Spot Noise - Texture Synthesis for Data Visualization. *Computer Graphics* 25, 4 (July 1991), 309-318.
- [18] ZABUSKY, N., AND SILVER, D. Case Study: Visualizing Clasical Problems in CFD. In *Proceedings of Visualization'92* (1992), A. Kaufman and G. M. Nielson, Eds., pp. 436-439.

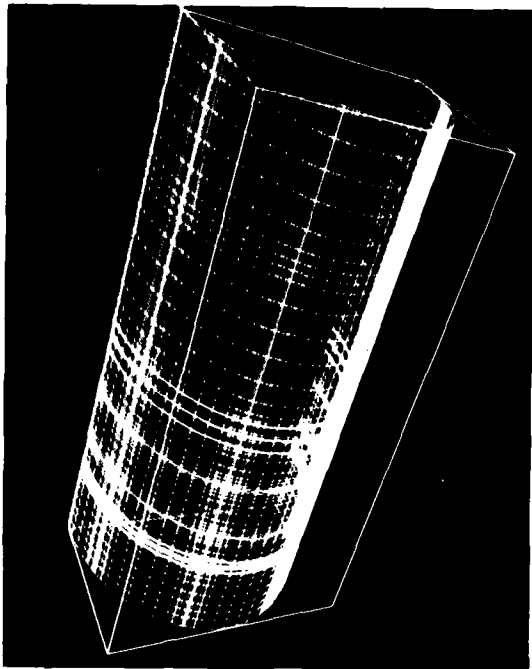


Plate 1: Traditional Particle Tracing.

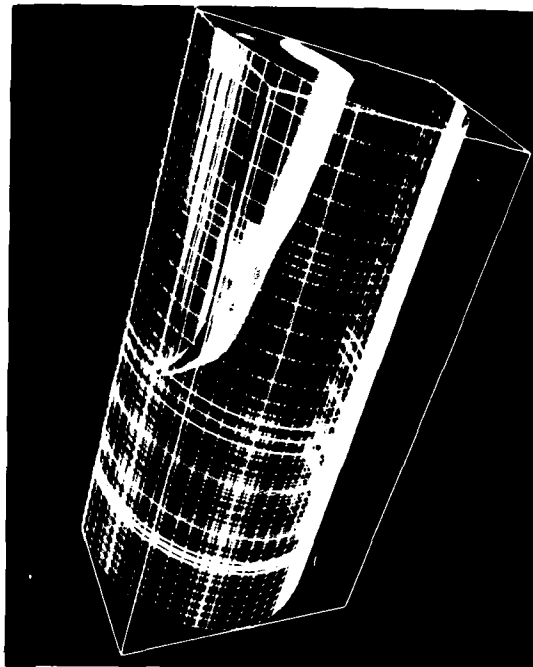


Plate 3: Cloud Tracing (Top View) using σ .



Plate 2: Cloud Tracing (Side View) using σ .



Plate 4: Cloud Tracing (Top View) using 2σ .

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 the 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 (Leave blank)		2. REPORT DATE August 1993	3. REPORT TYPE AND DATES COVERED Contractor Report	
4. TITLE AND SUBTITLE CLOUDING TRACING: VISUALIZATION OF THE MIXING OF FLUID ELEMENTS IN CONVECTION-DIFFUSION SYSTEMS			5. FUNDING NUMBERS C NAS1-19480	
6. AUTHOR(S) Kwan-Liu Ma Philip J. Smith			WU 505-90-52-01	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Institute for Computer Applications in Science and Engineering Mail Stop 132C, NASA Langley Research Center Hampton, VA 23681-0001			8. PERFORMING ORGANIZATION REPORT NUMBER ICASE Report No. 93-62	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Langley Research Center Hampton, VA 23681-0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA CR-191529 ICASE Report No. 93-62	
11. SUPPLEMENTARY NOTES Langley Technical Monitor: Michael F. Card Final Report				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category 61			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) This paper describes a highly interactive method for computer visualization of the basic physical process of dispersion and mixing of fluid elements in convection-diffusion systems. It is based on transforming the vector field from a traditionally Eulerian reference frame into a Lagrangian reference frame. Fluid elements are traced through the vector field for the mean path as well as the statistical dispersion of the fluid elements about the mean position by using added scalar information about the root mean square value of the vector field and its Lagrangian time scale. In this way, clouds of fluid elements are traced not just mean paths. We have used this method to visualize the simulation of an industrial incinerator to help identify mechanisms for poor mixing.				
14. SUBJECT TERMS scientific visualization, computational fluid dynamics, combustion engineering			15. NUMBER OF PAGES .16	
			16. PRICE CODE A03	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT	