(NASA-CR-158549) REVIEW OF COAXIAL FLOW GAS CORE NUCLEAR ROCKET FLUID MECHANICS Final Report (Illinois Inst. of Tech.) 84 p
HC A05/MF A01 CSCL 21F G3/20

N79-25132
Unclas
26531

FINAL REPORT

TO THE

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION

FOR GRANT NSG 7039

REVIEW OF COAXIAL FLOW GAS CORE NUCLEAR ROCKET FLUID MECHANICS

By

Herbert Weinstein
Professor of Chemical Engineering

Prepared by
Illinois Institute of Technology
Chicago, Illinois 60616

for OAST
Technical Program Monitors: F. C. Schwenk and K. Thom
July 1976

ABSTRACT

In an interrupted attempt to demonstrate the feasibility of using a coaxial flow gas core nuclear reactor as a rocket engine, NASA initiated a number of studies on the relevant fluid mechanics problems. These studies were carried out at NASA laboratories, universities, and industrial research laboratories. Because of the relatively sudden termination of most of this work in early 1973, a unified overview was never presented which demonstrated the accomplishments of the program and pointed out the areas where additional work was required for a fuller understanding of the cavity flow. This review attempts to fulfill a part of this need for the coaxial flow concept.

TABLE OF CONTENTS

INTRODUCTION

The concept of a gas core nuclear reactor appears to have first been discussed in the open literature about 20 years ago. The context in which it was discussed was as a rocket engine. The obvious advantages of the proposed engine was that the propellant could be heated to very high temperature , resulting in high specific impulse,and that high propellant flow rates could be achieved,resulting in high thrust levels. No other advanced concept proposed at that time promised this highly desirable combination of the requirements of manned, interplanetary travel.

The first concept of a gas core reactor was based on a vortex flow stabilization of the fuel region. Propellant entered the cylindrical cavity at the outer wall with a tangential velocity and flowed radially inward through the fuel region. It was anticipated that the fuel would remain suspended by the balance of shear and centrifugal forces. Heat transfer was to take place by conduction from the hot fuel directly to the propellant. However, early results on fuel containment were disappointing because of the rapid mixing and entrainment of fuel by the propellant. Two new GCR concepts were then advanced. Open cycle work was continued with the coaxial flow concept as its theme, and work was initiated on a closed cycle concept in which the fuel gas and propellant gas are separated by a transparent barrier.

The coaxial flow gas core reactor concept as first proposed was deceptively simple.[1] A low velocity inner stream of fissioning fuel gas was surrounded by a high velocity propellant stream. The hot inner gas transferred energy to the outer stream by thermal radiation, which was absorbed by the propellant because it was seeded with absorbing particles. In theory, at least, enough fuel could be fed into the cavity to maintain a critical mass. However, there were some economic limitations on the rate of fuel loss. The fluid mechanics studies which were undertaken to determine the rate at which fuel had to be fed into the reactor cavity to make up for the loss are the subject of this review.

The early fluid mechanics studies were begun before extensive nuclear studies had been made. In the earliest studies, it had not been apparent that the fuel had to be present in a large fraction of the cavity diameter and that the cavity length-to-diameter ratio had to be of the order of one. When it was realized that these conditions were necessary in order to keep the critical mass requirements within reason the fluid mechanics studies were redesigned to take into account the new picture of the cavity geometry. As work progressed further, it was seen that the flow was actually a turbulent, entrance region flow, almost totally dependent on the inlet configuration and conditions. The flow was transitional and unique and the usual assumptions of similarity and self-preservation were not valid for analytical studies, making necessary a large experimental program. Furthermore, in the actual reactor cavity, extremely large temperature and density variations uld exist and a fundamental understanding of the cavity flow was seriously lacking.

A program was set up by NASA to develop the basic understanding of the coaxial flow GCR fluid mechanics needed to determine concept feasibility. The program involved several laboratories after the initial concept formulation at Lewis Research Center. These were Lewis, United Aircraft Research Laboratories, Princeton University, and the Illinois Institute of Technology. Later, Aerojet Nuclear Company and the TAFA Division of the Humphreys Corporation were included. The experiments and analyses were carried out at the installation most suited for the type of work involved.

In January 1973, policy and budgetary factors forced the interruption of much of NASA's space nuclear program. Almost all of the fluid mechanics research in the coaxial flow GCR program ended abruptly. A unified overview, which demonstrated the accomplishments of the program and pointed out areas where additional work is still required, was never presented. This review is an attempt to fulfill a part of this need.

The objectives of this review are to establish the extent of our understanding of the coaxial fluid mechanics relevant to the cavity flow and to point out the important problems that are still to be solved. The additional objective of abstracting the entire literature of the program could not be included because it would make the review unwieldy in size. However, an extensive list of references is included. The recent publication of excellent reviews of fundamental work on relevant free turbulent shear flows[2] and coaxial free turbulent mixing[3] make it unnecessary to include a similar review in this work. The emphasis in this work is on discussion and comparison of studies of the factors which influence cold and hot flow containment. The work in these areas has not received any critical review in the past.

The papers discussed are limited to those from which the author feels a significant point can be made. The review is structured in such a way as to compare and contrast the related work of the program, rather than to preserve the chronological order of the work. Unfortunately, in such a structuring it often happens that a single work is only partially discussed without ever giving the whole of it. However, it is felt that the significant results of the program will be made more clearly visible with this approach and apologies are tendered to the authors whose works have been so treated.

FACTORS INFLUENCING COLD FLOW CONTAINMENT

In order to discuss the factors which influence containment of inner stream fluid in a confined coaxial flow, it is necessary to describe the various sub-regions of flow that exist along with their dominant characteristics. Figure 1 is a schematic of confined coaxial flow with the sub-regions labeled. Starting at the upstream center is the inner stream potential core. This region is dominated by inertial forces and pressure rather than shear forces. The shear forces do, however, control its size and for some cases the inner potential core may not exist. Surrounding the potential core is an annular mixing region which separates the inner and outer stream potential flows. It is a transitional shear layer with a rapidly growing turbulence level (for the typical turbulent case). It is dominated by the velocity difference between potential flows and the shapes of the inner and outer stream velocity profiles at the tip of the separating duct. The inner and outer stream turbulence levels and scales also have a moderate effect on the mixing region. The outer fluid potential flow is also dominated by inertial forces and pressure as well as by continuity effects due to the growing boundary layer on the confining duct. The annular mixing region thickens downstream to the point where it merges with itself and the inner stream potential core ends. At this point it can be said that the momentum defect of the trailing edge wake disappears. The flow downstream of this point is made up of at most, three regions. They are:the jetlike or wakelike (depending on velocity ratio) "free" shear flow region, the outerstream potential flow, and the duct boundary layer. Far enough downstream, the "free" shear flow region and boundary layer will merge and eventually fully-developed pipe flow will be approached in the duct.

The regions of interest in this paper are those upstream of the merger of the duct wall boundary layer and the "free" shear flow as contained by the dashed line in Figure 1. The main parameter for determining the nature of the flow is the velocity ratio of the streams. The Reynolds number has a second order effect and only on certain regions of the flow. The variation of velocity ratios can be broken up into three ranges for purposes of discussion.

When the velocity ratio is close to one, the predominant feature of the flow is the annular mixing region separating the potential flows. This mixing region is the wake from the trailing edge of the separating duct. Its spreading rate is affected by the initial velocity profiles in both streams at the trailing edge of the separating duct, the initial velocity difference across the region, and possibly by the initial turbulence levels in both streams. The effects of the initial turbulence would become more pronounced as the scales become of the order of the mixing region thickness or greater and possibly as the fluctuating velocity magnitude becomes of the order of the velocity difference or greater.

These effects are brought out in photographs of bromine-air coaxial flow experiments performed at Lewis Research Center.[4,5,6,7] The experiments were performed for a range of velocity ratios and inner and outer stream Reynolds numbers. At each set of values of the parameters, runs were made with honeycomb inserts in: a) neither duct; b) the inner stream duct; c) the outer stream duct; d) both ducts.

The experiments were not, however, well controlled. The insertion of the honeycomb sections, which had a passage width of about one-tenth of the inner duct diameter, changed free stream turbulence levels but also affected the boundary layers on both sides of the inner duct. This in turn affected the velocity difference across the mixing region because the flow rate ratio rather than the velocity ratio was actually held constant. The changes in the boundary layers with the addition of the honeycombs would be towards thinner boundary layers and smaller momentum defects. The former change would result in higher levels of vorticity and would tend to increase the rate of growth of turbulence in the shear layer while the latter change would decrease it. When factors increase the rate of growth of turbulence they will be called destabilizing factors because this study is only concerned with a very short region and a transitional shear layer may appear laminar over the very short axial length of interest. The change in free stream turbulence levels probably had the smallest effect on the apparent stability of the mixing region. In fact, examination of the photographs shows clearly that the scale of turbulence in the mixing region when the honeycombs ~~are~~ present is much too large for the turbulence to have resulted from the free stream levels alone. The well designed inlet section precludes large scale turbulence even without honeycombs present. The mixing region is very unstable in these cases, and the growth of turbulence appears to come from the typical shear layer growth mechanisms. The interaction between the initial width of the mixing region (the sum of the inner and outer boundary layers), the velocity difference across it, and the initial momentum defect is not well understood, and it is impossible to predict whether or not the gross effect of all the changes due to the introduction of honeycombs would be destabilizing or not for any given case.

The outer-to-inner stream velocity ratio range for which the honeycomb inserts seem to have their effect is between 0.2 and 2. The mixing region appears laminar for inner jet Re up to 2400 without either honeycomb section. With a honeycomb section in the inner stream duct only, the laminar appearance remains for almost all inner jet Re achieved, up to about 4000, but only for $u_o/u_i < 1$. This result is not readily explained on the basis of decreased inner stream turbulence as is implied in the study. A possible explanation is that the flat velocity profile with the same flow rate resulted in a smaller velocity difference across the mixing region. When a honeycomb insert was present in the outer stream alone, no effect of the honeycomb was apparent for $u_o/u_i < 1$, but there was a destabilizing effect for ratios greater than one. Since the outer stream transverse dimension is much larger than its boundary layer thickness, the outer stream honeycomb will have a negligible effect on the free stream velocity. It will, however, increase the vorticity levels in the outer stream trailing boundary layer and this is evidently destabilizing when the inner duct velocity is less than that in the outer duct.

When honeycomb inserts were present in both ducts, the laminar appearance persisted to inner jet Re greater than 2400 but less than 4000 for $u_o/u_i < 1$. However, for $u_o/u_i > 1$ the affect of both honeycombs was destabilizing as in the case with only outerstream honeycomb. The data for all four cases is presented in Figure 2 so that a comparison can be made.

It appears from these studies that when the inner stream is the faster one (the case of the primary jet), decreasing the true velocity ratio or decreasing the annular wake momentum defect is stabilizing. On the other hand, when the outer stream is the faster one (the case of a partially filled cylindrical wake), the vorticity level or boundary layer thickness is dominant in determining stability of the mixing region. Initial region turbulence levels appear to have very little, if any, effect.

The annular mixing region becomes of decreasing importance as the velocity ratio deviates significantly from one. As the velocity ratio becomes very small compared to one, the flow becomes that of a primary jet entraining a surrounding, secondary stream. Recirculation zones can be set up if the secondary stream can not meet the entrainment requirements of the primary jet. This recirculation zone is in the form of a ring pressed against the outer duct wall a few diameters downstream of the initial face. This range of velocity ratios is not the one of interest here and will not be discussed further.

The range of velocity ratios of interest for the coaxial flow GCR contains all those above a value of about 2 or 3. The most pertinent feature of this range is the solid cylindrical wake of the inner duct rather than the wake of its thin wall.

The wake is partially filled by the inner stream fluid, and for lower values of the velocity ratio the wake effect is partially mitigated. This can be thought of in terms of there being enough inner stream fluid to satisfy the entrainment requirements of the outer stream flow. However, as the velocity ratio becomes large enough, there is not enough inner stream fluid to satisfy the entrainment requirements of the outer stream and something similar to separation takes place with the development of radial pressure gradients and recirculating eddies behind the blunt cylinder end.[9] Figure 3 shows the development of the recirculation eddy as velocity ratio is increased. As the inner stream is entrained or accelerated along its outer edge, the streamlines move outward as they come closer together. Streamlines close to the centerline must diverge, as a result of the deceleration of the fluid, in order to satisfy the continuity condition. Thus, a region of positive pressure is generated on the centerline. If the pressure rise exceeds the dynamic head at the centerline, reversal of the flow takes place with the formation of a recirculation eddy. In this case, it is said that the inner stream cannot meet the entrainment requirements of the outer stream. If, however, the momentum of the inner stream is high enough, the accelerated outer region will reach the centerline before a complete reversal can take place. Here, the entrainment requirements are met.

The velocity is not the only factor in determining if recirculation will take place in a given coaxial flow. As has been mentioned before, the outerstream velocity profile is also very important. Its effect has been illustrated in terms of the flow induced upstream along the centerline by a cylindrical vortex sheet downstream of, and at the radius of, the inner duct.[9] This vortex sheet represents the boundary layers of the inner and outer streams. The thinner the dominant outerstream boundary layer is, the more "concentrated" its vorticity and the greater the strength of the vortex sheet. When the induced velocity, which increases with vortex sheet strength, reaches the magnitude of the inner stream velocity, recirculation is incipient.

A study in which the Navier-Stokes equations were solved numerically for the confined laminar coaxial flow system also showed the importance of the initial velocity profile in the development of recirculation eddies.[10, 11] Calculations were made with plug initial profiles as well as with parabolic initial profiles for a series of flow rate (average velocity) ratios. The recirculation always occurred at lower values of the flow rate ratio for the parabolic inlet profiles than for the plug inlet profiles. This is in apparent contradiction to the results of Masser[7] discussed above and emphasizes the complex interactions between the momentum defect, initial velocity profile, and velocity ratio on the inception of recirculation and on transition. The analysis was limited to laminar flow so that transition was not encountered, but it is important for flows with realistic Reynolds numbers. Another interesting conclusion made from the analysis which is related to transition is that results were presented which showed that recirculation was delayed to higher flow rate ratios with increasing bulk flow Reynolds number. The explanation was that since Re is the ratio of inertial to viscous terms, the viscous forces which resulted in entrainment became less important as Re increased, and enough inner stream fluid became available to satisfy the entrainment requirements of the outer stream. However, as Reynolds number becomes high enough, turbulence will develop and the effective ratio of inertial to shear forces, the turbulent Reynolds number, will become relatively very small. Recirculation would then again occur at much smaller velocity ratios. The experiment which was done in conjunction with this analysis showed that turbulence was actually present for laminar Reynolds numbers only slightly greater than the first value for which the delay in recirculation was seen from the calculations.

The containment of inner stream fluid can now be discussed in terms of the regimes and types of flow possible. It is obvious that recirculation will strongly enhance mixing and result in poor containment because the inner stream will be mixed internally with recirculated fluid and rapidly accelerated by and mixed with outer stream fluid, usually in a highly turbulent process. If recirculation is prevented from occurring, then the mixing takes place in the annular mixing region and the following cylindrical wake. The containment will be affected by the magnitude of the velocity gradients and the level of the turbulence. Suppressing the turbulence growth rate and maintaining small gradients will increase containment and these two conditions can usually be accomplished simultaneously.

It was recognized rather early in the program that large velocity gradients in the initial plane (narrow outer stream boundary layer on the trailing edge of the inner stream duct) resulted in a low value of containment and an analytical investigation was performed which demonstrated this. The investigation was on the effects of a momentum buffer region on the mixing and containment.[12] This investigation was performed before the existence and effects of recirculation eddies were understood and was concerned only with the effects of molecular and turbulent diffusion. It was found that there were optimum values of buffer region velocity ratio and thickness. The initial profiles of all three streams were taken as plug without zero velocity points at the ends of the separating ducts. The total outer stream flow remained constant as the fraction in the buffer region was varied. This latter condition caused the outer stream to inner stream velocity ratio to increase as the buffer region was made thicker. However, the general conclusion that decreasing the velocity gradients or vorticity increased containment was demonstrated for the restricted range of conditions studied in this work. It should be emphasized that in this study the eddy diffusivity was only a weak variable of the velocity difference change due to the buffer layer. Furthermore, the analysis could not include the gross changes in stability evident in the Masser study.[7]

In an experimental investigation[13] done at IIT in order to study the effects of free stream turbulence on the mixing, velocity and concentration profiles were measured for initial velocity profiles which were different from those in a previous investigation.[14] These profiles did not differ markedly, but it still can be seen from the results that when the initial outer-stream boundary layers in the second investigation were thicker, higher containment was the result. In this investigation, a screen was placed in the outer stream upstream of the initial face. This resulted in lower turbulence intensities in the outer stream, and a change in outer stream initial velocity profile due to tripping of the boundary layer on the outside of the inner duct. Results with the screen were compared to similar results taken with a boundary layer trip device which gave an outer stream profile similar to that with the screen but with higher turbulence intensities. Little difference in mixing and containment was seen due to the differing initial turbulence intensities. However, the velocity ratio was quite high in most of the cases of this investigation, and the solid cylindrical wake effect was the dominant one. For the cases where velocity ratios were high, the existence of a recirculation eddy was shown by moving a thread of cotton along the centerline and observing it being pulled upstream near the exit of the inner duct.

A series of experiments performed at UARL[15, 16, 17] was conducted to determine how containment could be improved by tailoring initial velocity profiles and turbulence levels. In a coaxial flow experiment, Scott industrial foam was placed across the entire duct at the inlet plane. It was shown that recirculation was suppressed up to very high velocity ratios. The effect of the foam is clearly seen in Figure 4. In frames a and b, no foam was present, and the inner stream, visualized with iodine, shows large scale turbulence on the edge and rapid mixing. In frames c and d, with foam present, it is seen that the inner stream has a smooth edge and maintains its integrity to a much higher velocity ratio. In a parallel investigation at IIT, shadow-graphs of a similar experiment were obtained and are shown in Figure 5.[18] Scott foam was placed at the initial face of the same apparatus used in reference 14 as shown in the first frame. Pictures were taken at several velocity ratios with and without a cone that produced, in a crude way, a buffer region between inner and outer streams. The inner stream was freon 12 and the outer stream was air. It can be seen here also, that the inner stream maintains its in-tegrity and shows smaller turbulence scales with the foam present and that the cone provides for a wider inner stream and a smaller scale of turbulence. What is most important in these two figures is that the inner stream decreases in width going through the foam.

The radial pressure gradient which is set up by the entrainment of inner stream fluid and the resulting high pressure region at the upstream end of the recirculation eddy when foam is not present is replaced by a radial inflow inside the foam. The initial profile at the downstream end of the foam is much smoother and monotonically increasing from the centerline to the beginning of the outer duct boundary layer. The velocity difference across the streams must be much lower than that upstream of the foam because of the radial inflow. Because this new initial profile has no minimum surrounded by inflection points, the shear layer generated turbulence is of much smaller scale and lower intensity. The effect of the foam on the free stream turbulence is probably of no importance at all. The major effect of the foam then is on the initial velocity profile, the initial velocity ratio, and the elimination of the radial pressure gradient. The work at UARL also included detailed measurements on a coaxial flow with a buffer layer and the Scott foam in a cavity which was carefully designed to minimize radial diffusive transport as well as radial pressure gradients. It was shown that containment could be increased dramatically over a two-stream system without foam.

The initial investigation on the spherical geometry with coaxial flow was carried out at Lewis Research Center.[19] The investigation was to determine if the spherical shape could be used advantageously for increasing containment. The flow experiment was carried out in a two-dimensional cavity with flat top and bottom and curved walls. The inner stream was smokey air introduced through a "shower-head" nozzle. Outer stream air entered all along the porous curved side walls with a radially inward directed velocity which was adjusted to be fairly uniform over the whole length of curved wall. The results were very encouraging. The flow appeared to be steady with a large volume of dense, smokey air in the middle of the cavity. Containment calculated on the basis of measured density of smokey air at the showerhead exit was very large at a flow rate ratio of 25. These results were understood to be of a very preliminary nature because it was a two-dimensional mockup of a three-dimensional flow and the Reynolds number of 1100 was quite low. However, they indicated that further work was warranted.

A series of experiments were then performed at UARL to determine containment levels in a spherical cavity.[20, 21] In an exploratory study, various types of fuel or inner stream injection configurations were studied separately and then the best design was used in an exploration for the optimal outer stream injection configuration. The spherical configuration for the "best" case containment studies is shown in Figure 6. The inner stream is injected through a porous sphere. The outer stream is injected at the top and bottom of the cavity with a tangential velocity component but the central section gives, essentially, only an axial component of velocity to the propellant. The reasons for this design being the "best" for spherical geometry is that it comes closest to the cylindrical case in design. Containment measured in the spherical geometries at UARL was always lower than that measured in cylindrical designs. The basic problem of the spherical design is in the radial pressure gradients which exist with curving streamlines. The tangential component of the outer stream at the top of the cavity sets up a radial pressure gradient with a pressure minimum at the centerline near the fuel injection point. However, at the downstream end of the cavity, the inward curving streamlines must begin to curve back out so that the fluid can exit the chamber through the nozzle. This reverse curvature causes a high pressure area at the centerline near the cavity exhaust. This high and low pressure system is very favorable for the formation of large recirculation eddies. These large eddies are visible in many of the flow visualization studies with spherical geometries. Furthermore, the pressure field set up directly around the fuel injection location due to the interaction of the gradients is very complex and can influence the inner stream fluid velocity profile at the injector surface. Finally, the entrainment necessary to "pull" the inner stream out to large radius against the induced pressure gradient probably also results in increased mixing of the fluids.

The conclusion that can be drawn from these containment studies at this time is that containment in cylindrical geometry cold flow studies is greatest when the radial pressure gradients are minimized and the initial velocity profiles are smooth, monotonic and have small velocity gradients. This combination of virtues has best been accomplished in the cylindrical coaxial flow experiments conducted at UARL. The factors affecting containment in spherical geometry cold flow are not as well understood. There is evidence, presented subsequently, that recirculation may be favorable to containment in this case. However, an almost cylindrical "spherical geometry" employing the features beneficial to containment in cylindrical geometries, appears to be one of the better possible designs.

COAXIAL FLOW AND CONTAINMENT

The containment by fluid dynamical means of the fissioning material in the cavity of the reactor is the key to the feasibility of gas core nuclear reactors. Containment, therefore, has been studied since the inception of the gas core program. Several different directions have been taken in these studies. The first of these, naturally, has been evaluation of the total fissionable material directly from the analytical solutions of the flow and mass transport equations. However, even the most detailed analyses undertaken were still lacking in some details of the actual flow system and experiments were performed to validate the analyses in terms of their ability to describe the proposed model rather than to represent the actual flow in the cavity. These analytical models were then used to generate enough information for parameter variation studies on fuel containment.

The other approaches to evaluation of fuel containment were of a directly experimental nature and usually were linked to studies concerned with the improvement of containment. Some of these studies dealt only with how a single feature of the flow would affect the containment of the fuel in order to further evaluate analytical approaches in which that particular feature was ignored.

The difficulties encountered in the evaluation of containment persisted throughout the evolution of the coaxial flow concept. The basic reason for the difficulties, as previously discussed, is the essentially square configuration of the cavity. Analytical methods are not available to treat the resulting turbulent entrance region flow with extremely large temperature and density gradients. Workers in the program could only approach a part of this difficult problem at a time, and the program was drastically scaled down in funding before all the pieces could be assembled.

Definition of Containment

The purpose of the containment calculation is to determine whether or not, or at what cost, a critical mass is contained in the reactor cavity. It is an interesting point that of all the problem areas of the cavity reactor concept, the nuclear one, that of calculating the required critical mass, is best understood. Criticality calculations can be made with reasonable accuracy to determine the mass required for criticality given the temperature, pressure, and location of the fuel in the cavity. Furthermore, critical experiments can be readily performed to check the validity of the calculations. The questions that must be answered from the fluid dynamics calculations concern the location of the fuel-rich region in the cavity and the rate at which fuel must be added to the fuel region to maintain a critical mass.

The initial concept of the geometry of the coaxial flow reactor was very simple. A cylindrical jet of gaseous fuel issued into a co-flowing stream of propellant. The flow was one consisting of concentric, almost cylindrical, stream tubes with the only deviation from cylindricity occurring because of mass continuity effects. Containment was defined, for this case, as the amount of inner stream fluid that is present in the cavity divided by the amount that would have been present had there been no acceleration of the inner stream. This containment factor may vary from a maximum of one to a minimum of u_i/u_o[22]

$$\bar{I} = \frac{u_i}{L} \int_o^L \frac{\int_o^R c r dr}{\int_o^R u c r dr} dz \qquad (1)$$

where subscripts i and o refer to initial properties of inner and outer gases respectively.

Another definition of containment which was used is the separation ratio, S.[23] S is defined as the ratio of the life of propellant particles to the life of fuel particles in the reactor cavity. For the original coaxial flow concept, this results in the simple expression

$$S = U_F/U_P \qquad (1a)$$

where U_F and U_P are the average velocities of the fuel and propellant flows,respectively. It can be shown that

$$S = \frac{u_i/u_o}{\overline{I}} \qquad \text{(1b)}$$

Thus, S can vary from a minimum of u_i/u_o to a maximum of 1, with the minimum value denoting the best possible containment.

As the coaxial flow concept evolved, the cavity shape became spherical. In this configuration, the radial velocity component was of the same order as the axial component and the definition of equation 1 was no longer a suitable measure of containment. Two definitions of containment then became accepted for reporting experimental results. One of these is the average partial pressure of fuel gas ~~divided~~ by the total pressure in the cavity,[17]

$$\frac{\overline{P}_1}{P} = \frac{\frac{1}{V}\int_V P_1 \, dV}{P} \qquad (2)$$

This can easily be converted to fuel mass,

$$\text{Fuel Mass} = \left(\frac{\overline{P}_1}{P}\right) \frac{PV}{RT} M \qquad (3)$$

This conversion of course is for uniform cavity temperature. For the case of varying cavity temperature, the conversion becomes

$$\text{Fuel Mass} = \int_V \frac{P_1 M}{RT} \, dV \qquad \text{(3a)}$$

This temperature effect is pointed out specifically because in hot flow studies discussed subsequently, mole fraction is plotted verses spatial position, and integration of mole fraction over the volume would not result in a measure of mass containment for these studies.

The other definition is called net volume fraction, and is the fraction of the cavity that would be filled if all of the fuel gas at its initial density was concentrated in one location,[24]

$$C.R. = \frac{\text{Total integrated density relative to 100\% for injection density}}{\text{total volume of cavity}} \tag{4}$$

This also can be readily converted to fuel mass,

$$\text{Fuel mass} = V\rho_{io}\ (C.R.) \tag{5}$$

These measurements of containment are more directly related to the mass of fuel present than is the first measure of equation 1.

Cylindrical Geometry Flow and Containment

The first analysis of the cylindrical coaxial flow in the GCR cavity used boundary layer equations, assumed constant pressure and laminar flow, and neglected the effects of the confining duct. This simplified the problem to the point that solutions could be obtained by a direct numerical integration without iteration starting from the initial velocity profiles.[25] Calculations showed that containment would be very high. This analysis was extended to turbulent flow by substituting eddy diffusivities for molecular diffusivities and used to interpret experimental results of a bromine-air coaxial flow.[26] The flow in the experiment was always turbulent except for one case and the data reported were the radial average bromine concentration normalized on the initial plane value as a function of downstream position. From this data, values of eddy diffusivity were obtained which brought into agreement the analysis and experiment. The containment found experimentally was not adequate for GCR feasibility and further exploration was undertaken to improve confinement. It is interesting to point out here, that what was not known at the time of this experiment was that the normalizing concentration of the initial face was almost always not that of pure bromine but represented a mixture of air and bromine due to the existence of the recirculation eddy at the exit of the inner duct. The actual containment, therefore, was even much poorer than was indicated for the high velocity ratio cases.

The analytical study on the effects of a momentum buffer region,which is discussed above, was then carried out at NASA Lewis.[12] The study showed the beneficial effects of tailoring the initial velocity profile within the limits of the analysis. Further studies on containment were then undertaken at NASA Lewis, United Aircraft Research Laboratories and IIT to determine how and to what level containment could be improved in the cylindrical geometry. All three studies were of the cold flow. The NASA Lewis studies were mainly of the flow visualization type and have been partially discussed above. The studies were pointed at an empirical determination of suitable entrance conditions for the cavity flow using honeycombs to lower inlet turbulence levels. Honeycombs were introduced into the upstream injection section for velocity ratios close to unity[4] and then for velocity ratios very different from unity.[5,7] As previously mentioned, the honeycombs did not improve containment for interesting values of velocity ratio. Downstream stagnation surfaces were also experimented with but no improvement in containment could be obtained with their use.[6]

The UARL studies were also experimental and were the first to use a short chamber to realistically mock up the cavity geometry. In the first study, the effects on containment of a buffer layer were determined.[15,16] Effects of other parameters such as velocity level, density ratio, injection radius ratio, cavity L/D and cavity outlet nozzle throat diameter were also investigated. The results indicated that the containment was most strongly affected by the occurrence of recirculation eddies in the inner jet region. For high values of the ratio of average outer-stream and buffer-stream inlet velocity to inner-stream inlet velocity, the recirculation occurred even though the buffer layer was present. This was not predicted by the study of Ragsdale[12] because the boundary layer equations employed could not permit a radial pressure variation. For flow conditions with a moderate value of this ratio (about 20 or less), a reduced level of turbulent mixing occurred between the three streams and the inner stream core extended to the exhaust nozzle. These results indicated that the buffer region was effective in keeping velocity gradients (or vorticity) low enough to prevent rapid transition and the generation of large scale turbulence which result in poor containment. It could not delay the gross wake effects from becoming important up to velocity ratios which were large enough to be of interest for the GCR.

In a second, exploratory study, Johnson introduced the idea of placing a porous material across the entire inlet plane downstream of the ends of the separating duct.[16,17] He used Scott industrial foams for this purpose. As discussed above, the foam permitted radial inflow without mixing of the streams, which essentially eliminated the radial pressure gradients which build up from the wake effect. The resulting flow just downstream of the foam had a reduced inner jet diameter and a reduced velocity ratio, but apparent containment with the foam was reported as being better than was obtained in the previous set of experiments without foam.

In the final study of this series, a number of cavity inlet configurations were tested, all with foam sections, in order to determine optimal containment. The inlet configurations included buffer regions, blocked inner sections of the inner stream entrance, blocked-off buffer region entrances, and various inner stream radius ratios. The experimental data obtained included velocity and turbulence intensity profiles and light absorption measurements which yielded concentration profiles. Figure 7 is a schematic of the test section used in the experiment and Figure 8 is a sketch of the inlet configuration. Two inlets of this type were used with a radius ratio of the inner stream equal to 0.6 in the first case and 0.7 in the second. The "best" inlet configuration had partially blocked inner stream inlets to spread the flow out enough so that the radial inflow in the foam would not make the emergent inner stream too small from critical mass considerations. The buffer region inlet was blocked completely, again to allow for radial inflow. Velocity profiles downstream of the foam are shown in Figure 9 for the radius ratio of 0.6 and inner and outer streams both of air. The weight flow ratio of outer to inner stream is 340, which corresponds to a superficial velocity ratio upstream of the foam of about 260. This is an extremely large value for a flow without recirculation. The velocity profiles downstream of the foam show a central region of essentially zero velocity and smoothly increasing velocity out to a peak and the beginning of the outer wall boundary layer. The maximum value of the ratio of peak outer stream velocity to the velocity at a radius ratio of 0.6 is only about 22, an order of magnitude smaller than 260. This change shows the combined effect of the radial inflow in the foam and the blocked regions at the inlet.

The containment values measured with this inlet at both radius ratios were the best ever reported for cold coaxial flow and are shown in Figure 10. Containment values are shown for both an air-air system and a Freon-12-air system which has a density ratio of 4.7 and which is about what would be expected with a uranium-hydrogen system and the GCR temperature distribution. For both flow systems containment in the GCR region of interest were obtained.

The series of studies performed at IIT were of a somewhat more fundamental nature. The first two of these were experimental investigations of turbulent mixing in an essentially unbounded coaxial flow[14] and in a shear layer.[27,28,29] It was determined that for the coaxial flow, wake flow effects dominated the high velocity ratio cases and that similarity of velocity profiles, density profiles and turbulent intensity profiles (self-preservation) was not evident until at least 3 inner stream initial diameters downstream. With the nuclear calculations indicating the need for a cavity L/D of about one and a large radius ratio of inner stream fluid, it became very evident that the cavity flow was an entrance region flow that would be difficult to treat analytically. Complete sets of velocity profiles, density profiles,and turbulence intensity profiles were presented for a range of velocity ratios and density ratios up to 7 to 1 for use in applicable theoretical studies. Another interesting result of these studies was that turbulence intensities in the region of interest increased with increasing velocity ratio as the outer stream velocity was held constant. This demonstrated that the turbulence which causes the rapid mixing is generated in the mixing region and not convected into the mixing region from upstream of the initial face.

A second study in the same coaxial flow test section demonstrated that upstream turbulence had little effect on the mixing,while the outer stream initial velocity profile had a very important effect on mixing. Further studies in this test section showed the turbulence intensities were somewhat damped when the inner stream density was increased to about 4 times that of the air outer stream[13] and explored the formation of recirculation eddies in the inner stream fluid.[9] This latter study showed the development of the radial and axial pressure gradients as the velocity ratio was increased. It also showed that for an unbounded coaxial flow,recirculation began at a velocity ratio of about 13 for a density ratio of one, and at about 26 for a density ratio of about 4, for the particular inlet configuration employed. While these results are unique to the test section in which they were measured, they demonstrate that recirculation is a function of both density ratio and velocity ratio and not simply of mass flux ratio. The data indicate that dependence is on the square root of momentum flux ratio, but there are not enough data available to verify this relationship.

Another experiment was also performed at IIT on the mixing of ducted, coaxial streams.[30] The inlet stream radius ratio was fixed at 0.6 and data obtained for velocity profiles, density profiles and turbulence intensity profiles. The recirculation patterns were again observed at the higher flow rates. Similarity of profiles, indicating a self-preserving turbulence, were not evident. Containment was evaluated for the cases with density ratio of 4 and showed a value of 0.1 for F_1/P at a velocity ratio of 29.2, a discouragingly low value. The profile data from this experiment was used in an analysis also conducted at IIT. The analysis was of a ducted coaxial flow of non-reacting fluids of different density.[31] It used the boundary layer equations as did those previously mentioned but took into account the outer duct shear stress effect by including an axial pressure variation. The solutions were obtained numerically and were fit to the experimental data to obtain coefficients for eddy transport. The solutions could be made to fit the data very well for low velocity ratios, but as the radial pressure gradients became significant, the analysis broke down. This indicated the necessity of using the full Navier-Stokes equations to describe the cavity flow.

A solution of the Navier-Stokes equations for a ducted coaxial flow was also carried out at IIT.[10, 11] The analysis was for a laminar flow of streams of different velocities, densities, and radius ratios. The bulk Reynolds numbers (based on total mass flow rate and outer duct diameter) was limited to less than 2000. The work explored the problems inherent to numerical solutions of the Navier-Stokes equations. It also showed explicitly the build up of radial and axial pressure gradients and the formation of recirculation eddies.

The outer duct was considered to be infinite in length in order to use the fully developed velocity profile as a downstream boundary condition so that the effects of a short L/D were not demonstrated. Because of the complex flow in the cavity, the use of a single value for turbulent Reynolds number is grossly inaccurate, so it is very difficult to relate the results such as the containment values calculated in this work to the more realistic turbulent case.

It should be pointed out in closing this section, that the gravitational force, or Froude number effect, while recognized by all of the investigators, had never really been separated from the mixing effects in discussing experimental data. The experiments were all performed with the test section axis parallel to the direction of the gravitational force. Acceleration due to the gravitational force would, when the inner stream density was substantially greater than that of the outer stream, have a considerable effect directly on the containment and on the inception of recirculation.

These effects were noted specifically by Bennett and Johnson[17] in some of their measurements. However, experimental containment data for the cylindrical cavity has never been corrected for this effect. The gravitational or body force field expected in the GCR had been evaluated analytically by Puttre [32] and considered more carefully in the spherical containment studies discussed below.

Spherical Geometry Flow and Containment

The investigation which initiated work on the spherical geometry was that of Lanzo.[19] He noted that because of pressure vessel design and wall cooling requirements, a spherical cavity shape would be more likely than a cylindrical one. In this spherical design the propellant would be introduced all along the cavity wall to satisfy wall cooling requirements and also to induce the inner stream to occupy a greater fraction of the cavity volume. An experiment was carried out to investigate the flow patterns which would occur in a spherical cavity with wall injection over the whole wall length. There were two objectives of the study. The first was flow visualization to gain qualitative information, and the second was to measure containment in order to determine whether or not further investigation of the spherical geometry was warranted. The flow experiment was, however, carried out in a two dimensional model as shown. This was done in order to have two flat transparent sides for the flow visualization and measurements. The apparatus consisted of these two sides held 6" apart by a 9" diameter circular sidewall which had a nozzle-like port. Opposite the port was a shower head inlet for smokey air. The sidewalls were porous so that the clear air which simulated the propellant entered the cavity with a radially inward directed velocity. This flow was held constant at 352 standard

CFH. The volumetric flow rate of the smoke-filled air which simulated the fuel was varied to give propellant-to-fuel flow rate ratios of 100, 75, 50, and 25. Three kinds of data were obtained. First, a shrouded hot-wire probe was used to measure incoming propellant velocity distribution around the curved cavity wall. Second, still and motion pictures were taken to obtain a qualitative description of the flow. Finally, desitometer traces of the still photograph negatives were made to obtain concentration profiles of the "fuel" in the cavity volume. The "fuel" concentration was referred to the value measured just downstream of the fuel injector.

The results of this study were felt to be very encouraging. After adjusting the propellant flow ratio so that it was fairly uniform over the whole length of cavity wall, the flow appeared to be steady with a large volume of relatively dense smokey air in the middle of the cavity. A typical case is shown in Figure 11. The containment calculations yielded a pure fuel volume of 0.51 at a flow rate ratio of 25 and of 0.28 at a flow rate ratio of 100. This measure of containment, as described above, is volume fraction which the fuel in the cavity would occupy if it were concentrated to 100% fuel and is equal to, for an isothermal cavity, the fuel partial pressure over the cavity pressure. These values are considered to be in the feasible range for the GCR.

These results were understood to be very preliminary and were intended only for determining whether or not additional work should be undertaken. The two-dimensional mockup of an axi-symmetric system is always crude, at best. One would expect more pronounced continuity effects and mixing in the axi-symmetric case. Also, boundary layer flows on the flat sides may have contained an unusually large fraction of the outer stream flow and measurements were not made to determine if this effect was present. Furthermore, the Reynolds number of 1100 in this experiment was very low, and the field was of uniform density.

Because these encouraging results left many unanswered questions, three new spherical geometry studies were initiated at UARL, Aerojet Nuclear Corp., and IIT. The intent was to obtain experimental data on a more realistic mockup (UARL), to develop the analytical capability, to study the flow in detail (IIT and UARL) and to start the experimental work in a location where the extension to a nuclear experiment could be undertaken. The two experimental investigations are discussed first.

Johnson at UARL[20] performed an exploratory study of the effects of "fuel" injection configurations and "propellant" inlet flow conditions in a spherical chamber. In these experiments, air was the propellant, or outer-stream gas, and air or Freon 11 was the inner or fuel gas stream. The inner stream was dyed with iodine for flow visualization. He designed an experimental apparatus in which both the upstream and downstream hemispheres could be modified to determine effects of propellant inlet flow configuration.

The inner gas or "fuel" injector configuration was studied in a coaxial flow apparatus previously discussed.[17] A portion of the outer gas entrance was blocked off over the injector to allow the inner gas to expand radially. Six different injector devices were tested. A ceramic filter cup provided for the proper axial and radial flow of inner stream gas. The flow from the filter cup was uniform, laminar like and did not have any jet characteristics.

Conclusions made concerning the inner-gas source region tests were that the inner-gas injector should provide for both radial and axial flow of inner gas, the outer flow should be blocked off upstream of the injector, and that it is possible to obtain a large volume of inner stream fluid with the proper outer-stream flow.

Studies of the outer-stream flow indicated that a porous wall injector which provided for an initially radial flow of the outer stream gave poor containment. An outer stream injection which was all tangential was experimented with and showed large recirculation cells and a large amount of mixing. Continued experimentation showed that containment was best when the outer stream was injected axially in the upper half of the cavity and tangentially in the lower half. However, containment was still not equivalent to that obtained with the cylindrical coaxial flow cavity. It was also found that the inner gas injector should be located about 1/10 of the cavity diameter downstream of the cavity top.

These experimental studies were continued in order to determine if containment could be substantially increased.[21] Outer stream injection studies were made in a wedge-shaped chamber and the complex dependence of containment on many flow parameters was demonstrated. The Reynolds numbers for the tests were high enough so that the flow was insensitive to this parameter. The most important parameters were the direction and distribution of outer stream velocity along the wall and the location of the inner stream source. These determined whether or not recirculation cells were set up and the location of the cell if it existed. As mentioned above, these parameters determine the pressure distribution in the cavity which in turn controls the recirculation. If the inner stream is entrained too rapidly, a high pressure region will exist near the inner stream inlet. The velocity direction near the cavity exhaust controls the strength of the downstream pressure maximum resulting from the streamlines bending back axially to go out the nozzle. These two pressure peaks, if high enough, will become stagnation points and cradle a recirculation eddy which usually causes a great deal of mixing of the two streams. The best configuration, in terms of containment, developed by Johnson is shown in Figure 6. It has a short tangential flow section up to the inner stream injector, a long axial flow section and a final tangential flow section. The first section helps to spread the inner stream,but the flow becomes axial in the second section before the inner stream can be entrained strongly enough to develop a very high pressure region just downstream of the inner-stream injector. The final tangential flow section is such as to make the radius of curvature of the streamlines in the cavity exit region fairly large in order to keep radial pressure gradients small.

The best containment results obtained after this long, experimental, trial and error process still were not as high as those obtained in the cylindrical coaxial flow experiments. Figure 12 shows the containment values plotted along with those of the previous work. It is seen that for both equal density cases and differing density cases containment is lower than with the cylindrical geometry.

It should be pointed out here that in this experiment with Freon-11 as the inner gas, the tests were run with the cavity exhaust located both at the top and bottom to demonstrate the effect of the gravitational forces. It was seen that in both cases the effect could be large. When the flow was upward, the body force tended to spread the inner stream and increase mixing, while when the flow was downward, the inner stream tended to accelerate and contract along the centerline. No quantitative measure of the body force field effect was obtainable, but the qualitative results indicated that it must be taken into consideration in the cavity design.

Containment was usually highest in this work, especially when the inner gas was of higher density than the outer gas, when a limited amount of recirculation was present. This helped to balance the gravitational forces acting on the heavier gas.

An extensive experimental investigation of spherical coaxial flow was carried out at Aerojet Nuclear Company.[24,33] The object of this work was to establish flow conditions which would be optimal from nuclear considerations. This was a first step in a program to run a flowing gas critical experiment which was never completed because of the termination of funding. Tests were conducted in two dimensional test sections with curved, louvered walls and flat sides and also in spherical test sections with louvered walls. Most of the tests, however, were done in tne two-dimensional test sections. The program was oriented towards obtaining a "working model", rather than a basic understanding of the cavity flow. Some scaling studies were done but no conclusive results were obtained.

The work included studies of the effects of inner stream injection, cavity shape, exhaust nozzle shape, outer stream injection configuration, density variation, and flow rate variation. Flow was metered outside the test section and the inner stream was dyed with oil smoke. Time exposure pictures were taken of the cavity and concentration of smokey inner stream fluid was obtained from densometric measurements of the film. The apparatus was designed primarily for ease in changing the various components and parameters rather than for making precise experimental measurements. Figure 13 shows tests with the two-dimensional test section, and Figure 14 shows tests with the spherical test section. After considerable experimentation it was concluded that containment at feasible levels could most probably be accomplished. Fuel volume fractions as high as 50% were reported for the two-dimensional test section and 33% for the spherical test section with flow rate ratios of 100 to 200. These results were for the equal density case and results with heavier inner stream fluid were substantially lower.

The optimal location of the inner stream injector and other configuration parameters were obtained for each test section in a trial-and-error procedure, but scaling laws for these parameters were not determined. However, scaling on reactor dimension was attempted using two test sections of each type, one twice as large as the other. Laminar Reynolds number and Froude number scaling was attempted, and it was found that since both of these parameters could not be held constant the scaling was somewhere in between matching the two but favoring the Reynolds number. However, matching the laminar Reynolds numbers will only match the inertial forces of two turbulent flows and not the turbulent shear forces. Since the containment results from a combination of inertial and shear forces, turbulence scaling must also be included and this may be the strongest parameter. Johnson, as reported above, found little variation in flow patterns with changing laminar Reynolds numbers.

Another conclusion of this work should be examined along with the results of Johnson. In the Aerojet work it was concluded that recirculation was very favorable for containment, while this conclusion was not so strongly shown in the UARL work. This is an interesting point and may be a result of the different velocity levels between the two investigations. The velocity level in the UARL experiments was about an order of magnitude higher than that in the Aerojet studies. The analytical study of Shavit, and Shavit and Lavan[10,11] for laminar flow agrees with the Aerojet conclusion by showing that for certain inlet flow configurations, the existence of a recirculation cell can improve containment. Apparently, the degree of turbulent mixing plays an important role in determining whether or not recirculation can be beneficial for containment. The gravitational forces also play a role in the overall balance. Recirculation diminishes the effects of the body force pulling the heavy, inner stream to the cavity exit, but if turbulent mixing is great enough, the net beneficial result of recirculation is lost because of the rapid entrainment of the inner stream along the edge of the recirculation zone.

A study of bouyancy or body force terms was also carried out along the lines of Puttre's work.[32] It was determined that increasing the density ratio always decreased containment level, but a scaling law for density variation effects could not be determined.

The concentration measurements made in the Aerojet study may have considerable inaccuracy. In Figure 13 three photographs are shown with different inner-stream injector positions. Both the inner-and outer-stream flow rates are the same for all three cases. Therefore, the mean, mixed concentration of smoke (flow averaged) must be the same in all three cases at the nozzle exit plane. However, the lowest figure shows a much greater average concentration of smoke exiting the cavity than the upper two. The effect seems to be due to how well the smoke is distributed in the direction transverse to the plane of the film. In the figure the smokey air seems to be concentrated along the axis of the test section and this indicates that a considerable variation in measured containment would result from variations transverse to the film plane and caused by two-dimensional flow regions such as the boundary layers on the flat surfaces. This, and other sources of inaccuracies with a concentration measurement of this type indicate, that while the trends reported in this work are correct, the actual values of containment reported should be taken as being very approximate.

The analytical investigations of spherical coaxial flow were both numerical solutions of the Navier-Stokes equations. Johnson used a modified version of the Basic Computer Program of Gosman, et al[34] for determining streamline distribution for applicable boundary conditions. The modification of the program was done to accommodate the type of boundary conditions pertinent to the GCR cavity. The turbulent flow in the cavity was simulated by using a single very large value of the kinematic viscosity. Results were obtained which showed the way the strength and location of the recirculation zone varied with outer-stream and inner-stream inlet configurations and with varying density ratios. The value of the gravitational constant was also varied to demonstrate its effect on the flow field. Figure 15 shows the streamlines and isodensity lines for two values of initial density ratio ρ_i/ρ_o for a given set of boundary conditions, and illustrates the effect of varying this parameter. The calculated streamlines and isodensity lines have the same general characteristics observed experimentally in the tests reported on previously.

A second numerical solution of the Navier-Stokes equations was partially completed at IIT before funding was terminated.[35] This work was directed at the development of a new computer program that would be directly applicable to the spherical geometry of the GCR cavity. Some preliminary results were obtained for laminar flow and a density ratio of one which showed how the recirculation was affected by changes in outer stream injection configuration.

These analytical tools can provide a very useful and inexpensive means of eliminating the major part of experimental trial-and-error optimization. Good approximations of inlet flow geometry and cavity shape can be found with "computer experimentation" and with only final optimization requiring laboratory experimentation. Scaling laws can also be determined from numerical solutions with verification in the laboratory. The present computer programs must, however, be developed further to include the effects of turbulence, energy generation, temperature variation, and the proper cavity geometry in order to become a useful tool.

EFFECTS OF HEAT GENERATION ON CONTAINMENT

Most of the studies done on the containment of fissionable material of fuel in the cavity considered only the cold flow of gases. The effects of the very large heat generation rate in the fuel gas were neglected because of the difficulties encountered in treating them both analytically and experimentally. The few studies that have been performed on "hot flows" do not readily yield values for containment but rather indicate how the heat release will affect the containment as it is determined from cold flow studies.

The experimental work done on hot flows includes a series of investigations by Grey and coworkers at Princeton University on a flowing argon plasma and cold coaxial stream of several different gases.[36, 37, 23] Cold flow data were also taken for comparison. Their work also included an analysis of coaxial flow mixing. Both laminar and turbulent flows were investigated. The earlier work centered on the laminar case, but results showed that a laminar flow could not be maintained in their apparatus. In the turbulent flow studies maps of concentration, velocity and temperature were obtained with a cooled probe technique. The flow in this system was dominated by a wake effect, which the investigators attempted to quantify, and high inlet turbulence level. Schlieren pictures of the flow field showed that the coaxial field fluid which issued from a section of tubes maintained its individual jet stratification for at least one inner jet diameter downstream, inhibiting mixing to some extent. However, the wake effect was, in general, so strong that mixing occurred very rapidly. The important conclusions that were drawn from this work are that the effects of core temperature on the concentration profiles are not large and that containment of argon within a cylinder defined by the argon inlet duct diameter is essentially complete. Furthermore, the spreading rate of the argon appears to be less with the hot flow cases than in similar cold flow cases. These conclusions are stated here because they are in disagreement with published results from a later series of arcjet studies.

This later series of hot flow studies were done at the TAFA division of the Humphries Corp.[38,39,40,41,42,43,44] TAFA's objective was to develop an induction heated plasma simulation of a gas core reactor. Data consisting of concentration, temperature and stagnation pressure measurements[40] were taken in a coaxial flow of argon plasma and air or hydrogen. The plasma was induced by a radio-frequency field after the arc was started up with a D.C. discharge. The air/argon mass flow ratios used were 0.6, 5.55 and 19.7 and the hydrogen/argon mass flow ratio was limited to one value of 1.67. Both hot flow and cold flow runs were performed and it was concluded from a comparison that "the plasma eliminates all turbulent recirculation present in the cold flow and appears to behave as if it had a definite skin or boundary similar to a gas/liquid interface." Some of their hot and cold flow concentration data is reproduced in Figures 16-18. Figure 16 shows concentration measurements for cold, laminar, axial flow of argon and air. Both streams have equal volumetric flow rates of 100 SCFH. The parabolic lines of constant mole fraction are cited as evidence of purely laminar flow. The strongest evidence presented by this figure, however, apparently invalidates the concentration measuring scheme employed. The mean mixed mole fraction of air for this case is 0.50. Yet, at two diameters downstream, the parabolic flow averaged mole fraction of air is at least 0.70. However, at this axial station the velocity profile is probably closer to plug than parabolic, and a plug profile averaging would give a mean, mixed mole fraction of air of about 0.8. This indicates that concentration measurements are accurate to, at best, about $\pm$ 50%. Other data are presented for cold and hot flows successively, for different mass flow rate ratios. In these cases there is a tangential or swirl component of the velocity of the argon to stabilize the arc.

The discrepancy between the measured concentrations and the mean mixed mole fraction exists for each of the mass flow ratios investigated. The equal velocity case, $W_{air}/W_{argon} = 0.67$, is shown in Figures 17 and 18 for the cold flow and hot flow, respectively. In the cold flow case, at 2" downstream the air mole fraction is shown as 0.5 at the centerline. The region off the centerline must have considerably higher mole fractions of air, yet the mean mixed mole fraction of air is 0.451, according to a mass balance. Another peculiar feature of the TAFA results is visible in Figure 18 for the hot flow. At the initial plane of the mixing region the argon concentration ranges as high as 0.50 in the entering air stream. This peculiarity exists in most of the remaining data for both hot and cold flow.

One possible explanation for the disagreement can be based on the measuring device employed. A sample of fluid was removed from the flow field with an aspirating probe. If the probe sucked in fluid at a rate higher than the local flow rate in the field, it would perturb the flow field considerably and result in the measurement of a concentration averaged over a volume quite large compared to the probe tip.

Another observation that can be made from the cold flow figures is that the inner stream spreads rapidly, radially outward, past the radius of the argon duct. This is in direct disagreement with essentially all other cold coaxial flow experiments with the outer stream to inner stream velocity ratio substantially greater than one. The Princeton hot flow studies, which also had a swirl component to stabilize the arc, show a decrease in spreading rate from the cold flow to hot flow experiments as do the TAFA experiments, but the Princeton measurements also conclude that the argon does not spread beyond the inner duct radius. Furthermore, the Princeton plasma is being cooled, so the spreading rate changes should be different than in the TAFA work where energy is being put into the plasma by the RF field. These apparent inconsistencies and disagreement with previous work make conclusions on containment from the TAFA work of doubtful value.

A second conclusion drawn from the TAFA work is that the plasma has a "hard skin". This is based on high speed photographs showing 250 micron tungsten particles "bouncing off the plasma fireball." It is not clear that this conclusion has ever been confirmed by reference to observations made by other investigators in the field of plasma physics working with arcs which are not in RF fields. An alternate possible explanation of the bouncing relates to the Leidenfrost phenomenon which explains why a drop of water will bounce off a hot frying pan. The tungsten particle would begin to vaporize as it entered the very hot plasma region. The high rate of vapor generation would then drive the particle back out of the arc.

It is also true that electrostatic fields exist in the plasma due to the RF field. Calculations of the electrostatic force required to balance the gravitational force on the particle indicate a charge equivalent to 10^3 to 10^4

electrons on the surface of the tungsten ball are required. It is possible that a charge of this magnitude could be accumulated by the tungsten as it passes through the outside fringe of the plasma since tungsten would tend to "poison" the arc. These alternative explanations for the bouncing particles are also not conclusive. However, the existence of a "hard plasma skin" should be left open to question until it is confirmed in a plasma without an RF field.

In a later report covering tests which were carried out by TAFA on a curved permeable wall induction torch, the plasma was maintained by vaporizing solid material which ionized.[41] After startup, the plasma was fed by the vaporized solid with no through-put of ionizing gas. The solid to coolant gas mass flow ratio was between 1/1500 to 1/5000. Because of the very different processes taking place, these numbers cannot be taken to have any relationship to containment requirements in a cavity reactor.

In this report, experiments were also carried out with a flowing argon plasma and with a coolant gas flow, all of which entered the flow field radially through a porous wall. In these tests, as in the previous ones, the hot flow conditions were said to prevent all recirculation due to wake effects. This can be explained as being due to the rapid expansion of the argon stream which in turn is due to its rapid temperature rise. In fact, a gas experiencing a temperature change from 300°K to 9000°K would expand in volume by a factor of 30. Since the confining walls prevent radial expansion of the argon, the axial velocity of the argon increases by at least a factor on the order of 30. In all the tests performed at TAFA, except one, the initial (cold) velocity ratio was about 34 or less, and the exceptional case was about 45. If these ratios are divided by 30, it is seen that actual velocity ratios is the plasma region are order one or less, so that the strong wake effect which causes recirculation is just not present in the hot flow cases. This axial expansion of the inner stream is also apparent in the Princeton work where velocity traverses are presented.[23] However, the incomplete data sets make it difficult to demonstrate it quantitatively. Figure 19 shows a case with initial velocity ratio of about 1 where initial velocities are based on initial plane average conditions. However, the centerline temperature at the initial plane of the mixing region is 6000°R, and the centerline velocity is almost 3 times the coolant gas velocity. The centerline temperature decays to 3000°R in two diameters, and the centerline velocity falls to about half of the initial value, indicating expansion and contraction take place almost entirely in the axial direction for these experiments where the outer fluid was not confined.

An analytical study of the effects of heat generation on an entrance region coaxial flow was performed at IIT.[45]

In this study, the equation set was formulated to represent the gross behavior of the flow field and to avoid the complexity of a more detailed analysis. The field was modeled as a laminar, confined coaxial flow for which the boundary layer assumptions are valid. Only radial radiative transport of energy was considered,and the two fluids were assumed to be immiscible. The results for one typical case are shown in Figure 20. Here z is axial distance made dimensionless on outer duct radius times Reynold's number, and α is the location of the interface between the two immiscible streams. Subscript α is the value of the inner stream property at the interface.

The centerline velocity changes from 1/10 the outer stream value at the initial plane, to 1.4 times the initial value of that of the outer stream. The centerline temperature increases very rapidly and then actually falls off slightly as the thermal radiation becomes dominant. It is also seen that the interface location falls rapidly from 0.7 to about 0.3 as the centerline velocity increases by a factor of 14, and centerline temperature increases by a factor of only 3. If the velocity increase was due only to thermal expansion axially along the centerline, the velocity change there would be only by about a factor of 3. However, all of the inner stream expands due to heat generation and the energy radiated to the outer stream causes expansion of the outer stream also, and all of this fluid expands radially inward to some extent, because of the confining wall. This effect, coupled with the buildup of the wall boundary layer accounts for the additional increase in centerline velocity. This type of behavior is indicated in the TAFA results for their hot flow cases (Figure 18). In all their cases, the lines of high constant mole fraction of argon stretch out to a considerable extent in a narrow region along the centerline. This is consistent with the idea of a radially shrinking, axially expanding core of argon.

There is, at present, one investigation still being carried out at Princeton University.[46] It is a numerical evaluation of heat transfer in a fissioning uranium plasma core reactor cavity operating with seeded hydrogen. The heat transfer in the cavity is considered in great detail, but at the present stage of analytical development flow patterns in the cavity are assumed, so that thermal effects on the flow cannot be evaluated.

It is difficult to draw conclusions from the relatively small amount of work done on how the internal heat generation affects containment. In the discussion of the previous section, it was seen that cylindrical coaxial flow gave the best cold-flow containment found to date. However, the cylindrical boundary causes large axial acceleration when internal heat generation is present. It is possible then, that a spherical cavity outer boundary may be desirable for containment with heat generation. The key to success would be to use the thermal expansion to take place radially instead of axially, while mixing and entrainment are minimized by inducing axial streamlines downstream of the expansion.

SUMMARY

The accomplishments of a project-oriented research program are difficult to describe when major elements of the program are interrupted for several years. It is natural in such a program to progress from stage to stage without fully unifying and documenting the work of each stage because it appears at the time wasteful of time and funds to do so. However, the full retrospective documentation of program accomplishments, when the program has not been carried to a logical conclusion, is almost impossible.

A large number of fluid mechanics studies of the coaxial flow in the GCR cavity have been carried out. Some of this work has significance beyond the program itself, and some of it was so restricted in scope that it has value only toward the development of a coaxial flow GCR. This type of discrimination was not undertaken as a part of this review. This review attempts to describe what is presently known about the flow in the cavity. It is obvious that feasibility cannot be evaluated today.

It has been shown that cold flow containment levels in a cylindrical cavity are at least into the marginal range, and further improvement is possible. Some experimental evidence exists which indicates cold flow containment in a spherical cavity has been brought to the same level. There are also experiments which indicate that temperature effects will be favorable, but these have not been adequately documented. The development of analytical tools for studying the flow has not been carried to the stage at which these tools become

truly useful. What is apparent from the review is that even though much has been done to demonstrate feasibility, a great deal more still needs to be done.

The general nature of the flow in both cylindrical and spherical cavities is known. The effects of the single variation of each of the most important flow parameters are also understood. However, neither an analytical nor an experimental model has been developed which describes the physical situation in enough detail to provide an accurate assessment of fuel containment. Such models would include means for simulating and varying velocity profiles, density profiles, and turbulence levels along an entire, arbitrary, cavity boundary and a fuel injection device. They would also include a means of simulating the heat generation and transport in the cavity, and of simulating gravitational effects.

In order to continue the program to demonstrate coaxial flow GCR feasibility, further work must be done on both analytical and experimental models of the cavity flow. These models must be more quantitative than those produced in the past. The present state of the art of modeling of the cavity flow is such that development of a realistic analytical model can be started immediately. However, because of conflicting results and inaccuracies of some of the previous experimental work with high temperature flows, the experimental modeling should not be started at a stage as advanced as that of the analytical modeling. Rather, the development of a non-nuclear experimental facility should be started with the initial objective of repeating the hot plasma-cold propellant coaxial flow studies in both cylindrical and spherical geometries. Specific questions such as those concerning mixing with large velocity ratios based on hot flow densities and turbulent transport across a large temperature gradient should be addressed in the early work with this proposed facility.

It should be pointed out in closing that the feasibility of any given concept is to some extent a function of the value of the conceived device. Therefore, any continuing work on GCR feasibility should be broad enough in scope to be relevant to the wide range of possible applications of the device.

References

1. Weinstein, H. and Ragsdale, R. G., "A Coaxial Flow Reactor - A Gaseous Nuclear Rocket Concept," American Rocket Society, Preprint 1518-60, 1960.

2. Free Turbulent Shear Flows, Volume 1 - Conference Proceedings, NASA SP-321, 1973.

3. Harsha, P. T., "A General Analysis of Free Turbulent Mixing," AEDC-TR-73-177, May, 1974.

4. Ragsdale, R. G. and Edwards, O. J., "Data Comparisons and Photographic Observations of Coaxial Mixing of Dissimilar Gases at Nearly Equal Stream Velocities," NASA TN D-3131, 1965.

5. Taylor, M. F. and Masser, C. C., "Photographic Study of a Bromine Jet in a Coaxial Airstream," NASA TN D-4660, 1968.

6. Masser, C. C. and Taylor, M. F., "Photographic Study of a Bromine Jet Flowing in a Coaxial Airstream and Impinging on a Stagnation Surface" NASA TN D-5209, May, 1969.

7. Masser, C. C., "A Photographic Study of a Bromine Jet in a Coaxial Airstream with Honeycomb at the Injection Plane," NASA TM-X-1900, October, 1969.

8. Barchilon, M., and Curtet, R., "Some Details of the Structure of an Axisymmetric Confined Jet with Backflow," J. of Basic Eng., 86, P777 (1964).

9. Rozenman, T., and Weinstein, H. "Recirculation Patterns in the Initial Region of Coaxial Jets" NASA CR-1595, May, 1970.

10. Shavit, G., "Analytical and Experimental Investigations of Laminar Mixing of Homogeneous and Heterogeneous Jets in a Confining Tube", Ph. D. Thesis, Illinois Institute of Technology, June, 1970.

11. Lavan, Z., and Shavit, G., "Recirculation Patterns in Confined Laminar Jet Mixing" Israel J. of Tech., 9, 51, (1971).

12. Ragsdale, R. G., "Effects of a Momentum Buffer Region on the Coaxial Flow of Dissimilar Gases" NASA TN D-3138.

13. Kulik, R. A., Leithem, J. J. and Weinstein, H., "Effect of Free Stream Turbulence on Coaxial Mixing," NASA CR-1336, May, 1969.

14. Zawacki, T. S., and Weinstein, H. "Experimental Investigation of Turbulence in the Mixing Region Between Coaxial Streams," NASA CR-959, Feb., 1968.

15. Johnson, B. V. "Experimental Study of Multi-Component Coaxial-Flow Jets in Short Chambers," NASA CR-1190, April, 1968.

16. Johnson, B. V., "Exploratory Experimental Study of the Effects of Inlet Conditions on the Flow and Containment Characteristics of Coaxial Flows," NASA CR-107051, 1969.

17. Bennett, J. C., and Johnson, B. V., "Experimental Study of One-and Two- Component Low-Turbulence Confined Coaxial Flows," NASA CR-1851, June, 1971.

18. Kulik, R. A., and Weinstein, H. Private Communication, 1969.

19. Lanzo, C. D., "A Flow Experiment in a Curved-Porous-Wall Gas-Core Reactor Geometry," NASA TM-X-1852, August, 1969.

20. Johnson, B. V., "Exploratory Study of the Effects of Injection Configurations and Inlet Flow Conditions on the Characteristics of Flow in Spherical Chambers," NASA CR-1965, January, 1972.

21. Johnson, B. V., "Experimental and Analytical Study of One and Two Component Flows in Spherical Chambers," NASA CR-2282, August, 1973.

22. Weinstein, H. and Todd, C. A., "Analysis of Mixing of Coaxial Streams of Dissimilar Fluids Including Energy - Generation Terms," NASA TN D-2123, March, 1964.

23. Williams, P. M. and Grey, J., "Simulation of Gaseous Core Nuclear Rocket Mixing Characteristics Using Cold and Arc Heated Flows," NASA CR-690.

24. Kunz, J. F., Suckling, D. H., and Cooper, C. G., "Phase I Topical Report - Flowing Gas, Non-Nuclear Experiments on the Gas Core Reactor," NASA CR-120824, Feb., 1972.

25. Weinstein, H. and Todd, C. A., "A Numerical Solution to the Problem of Mixing of Laminar Coaxial Streams of Greatly Different Densities - Laminar Case," NASA TN D-1534, 1963.

26. Ragsdale, R. G., Weinstein, H. and Lanzo, C. D., "Correlations of a Turbulent Air-Bromine Coaxial-Flow Experiment," NASA TN-D-2121, 1964.

27. Baker, R. L., and Weinstein, H., "Analytical Investigation of the Mixing of Two Parallel Streams of Dissimilar Fluids," NASA CR-956, January, 1968.

28. Baker, R. L., and Weinstein, H., "Experimental Investigation of the Mixing of Two Parallel Streams of Dissimilar Fluids," NASA CR-957, January, 1968.

29. Baker, R. L., Rozenman, T., and Weinstein, H., "Calculation of Stability of Shear Flow with Density Gradient and Viscosity," NASA CR-958, February, 1968.

30. Leithem, J. J., Kulik, R. A. and Weinstein, H., "Turbulence in the Mixing Region Between Ducted Coaxial Streams," NASA CR-1335, July, 1969.

31. Ghia, K., Torda, T. P., and Lavan, Z., "Turbulent Mixing of Heterogeneous Axisymmetric Coaxial Confined Jets," NASA CR-1615, May, 1970.

32. Puttre, H. A., "Effect of Bouyancy on Fuel Containment in an Open-Cycle Gas-Core Nuclear Rocket Engine," <u>2nd Symposium on Uranium Plasma: Research and Applications</u>, Atlanta, Ga., November, 1971.

33. Kunze, J. F., Cooper, C. G., and Macbeth, P. J., "Phase II Topical Report - Flowing Gas, Non-Nuclear Experiments on the Gas Core Reactor," NASA CR-121191, June, 1973.

34. Gosman, A. A., Pun, W., Runchal, A. K., Spaulding, D. B., and Wolfstein, M., <u>Heat and Mass Transfer in Recirculating Flows</u>, Academic Press, New York, 1969.

35. Lavan, Z., Illinois Institute of Technology, Private Communication, 1973.

36. Grey, J., William, P. M., and Fradkin, D. B., "Mixing and Heat Transfer of an Argon Arcjet with a Coaxial Flow of Cold Helium," NASA CR-54438, October, 1964.

37. Grey, J., Sherman, M. P., Williams, P. M., "Laminar Arcjet Mixing and Heat Transfer: Theory and Experiments," AIAA J., 4, p. 986, (1966).

38. Thorpe, M. L., "Induction Plasma Heating: System Performance, Hydrogen Operation and Gas Core Reactor Simulator Development", NASA CR-1143, August, 1968.

39. Thorpe, M. L. and Scammon, L. W., "Induction Plasma Heating: High Power, Low Frequency Operation and Pure Hydrogen Heating", NASA CR-1343, August, 1969.

40. Dundas, P. H., "Induction Plasma Heating: Measurement of Gas Concentrations, Temperatures, and Stagnation Heads in a Binary Plasma System", NASA CR-1527, February, 1970.

41. Vogel, C. E., "Curved Permeable Wall Induction Torch Tests:, NASA CR-1764, March 1971.

42. Poole, J. W., and Vogel, C. E., "Induction Plasma Nozzle Tests", NASA CR-1765, April, 1971.

43. Vogel, C. E., Poole, J. W. and Dundas, P. H., "Radiation Measurements and Low Frequency and High Pressure Investigations of Induction Heated Plasma," NASA CR-1804, May, 1971.

44. Poole, J. W. and Vogel, C. E., "Induction Simulation of Gas Core Nuclear Engine", NASA CR-2155, February, 1973.

45. Bobba, G. K. M., "Laminar Confined Coaxial Flow with Heat Generation," Ph.D. Thesis, Illinois Institute of Technology, December, 1974.

46. Smith, D. E., Smith, T. M., and Stoenescu, M. L., "Heat Transfer Evaluation in a Plasma Core Reactor," Proceedings of Princeton University Conference No. 133, on Partially Ionized Plasmas including the 3rd Symposium on Uranium Plasmas, June, 1976.

47. Ragsdale, R. G., "Some Fuel Loss Rate and Weight Estimates of an Open Cycle Gas-Core Nuclear Rocket Engine," NASA TM X-52775, 1970.

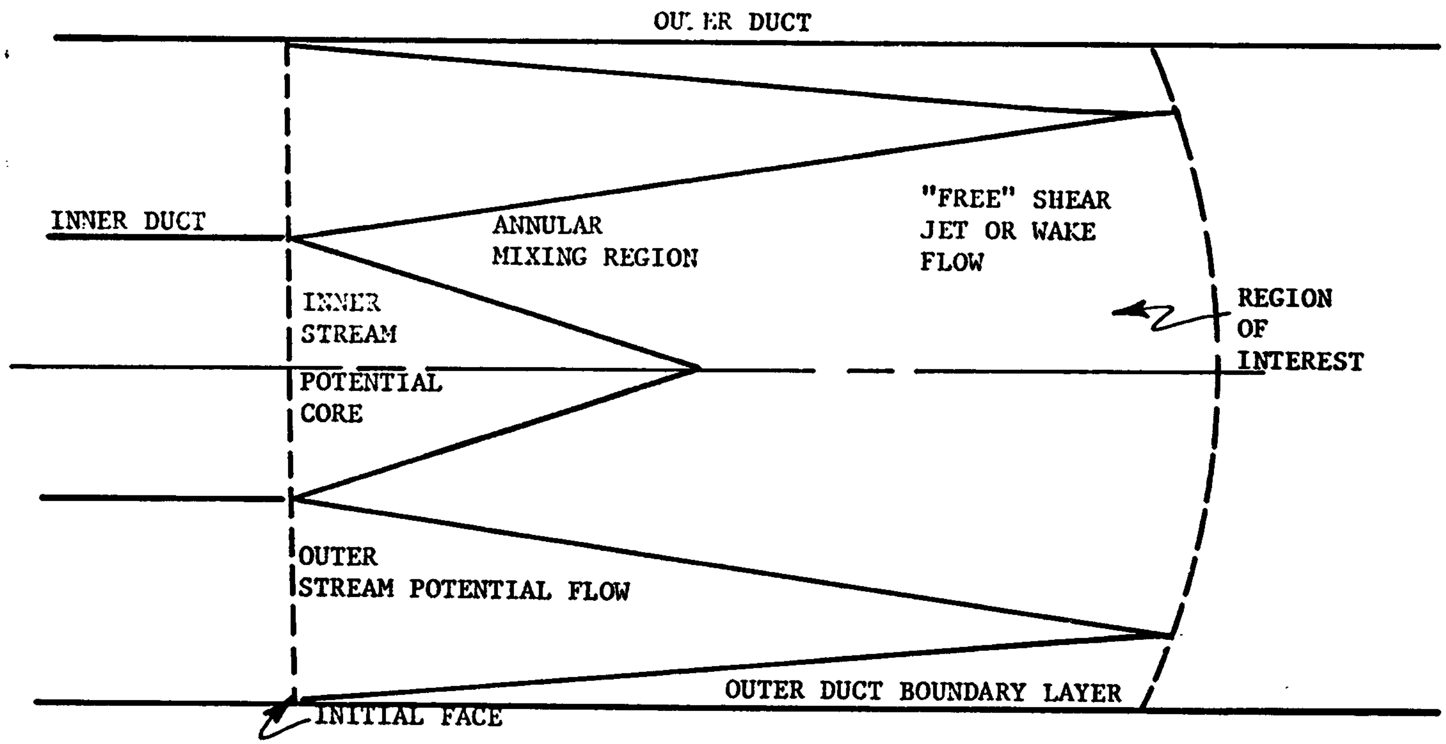


FIGURE 1. Schematic of Flow Regimes in Ducted Coaxial Flow

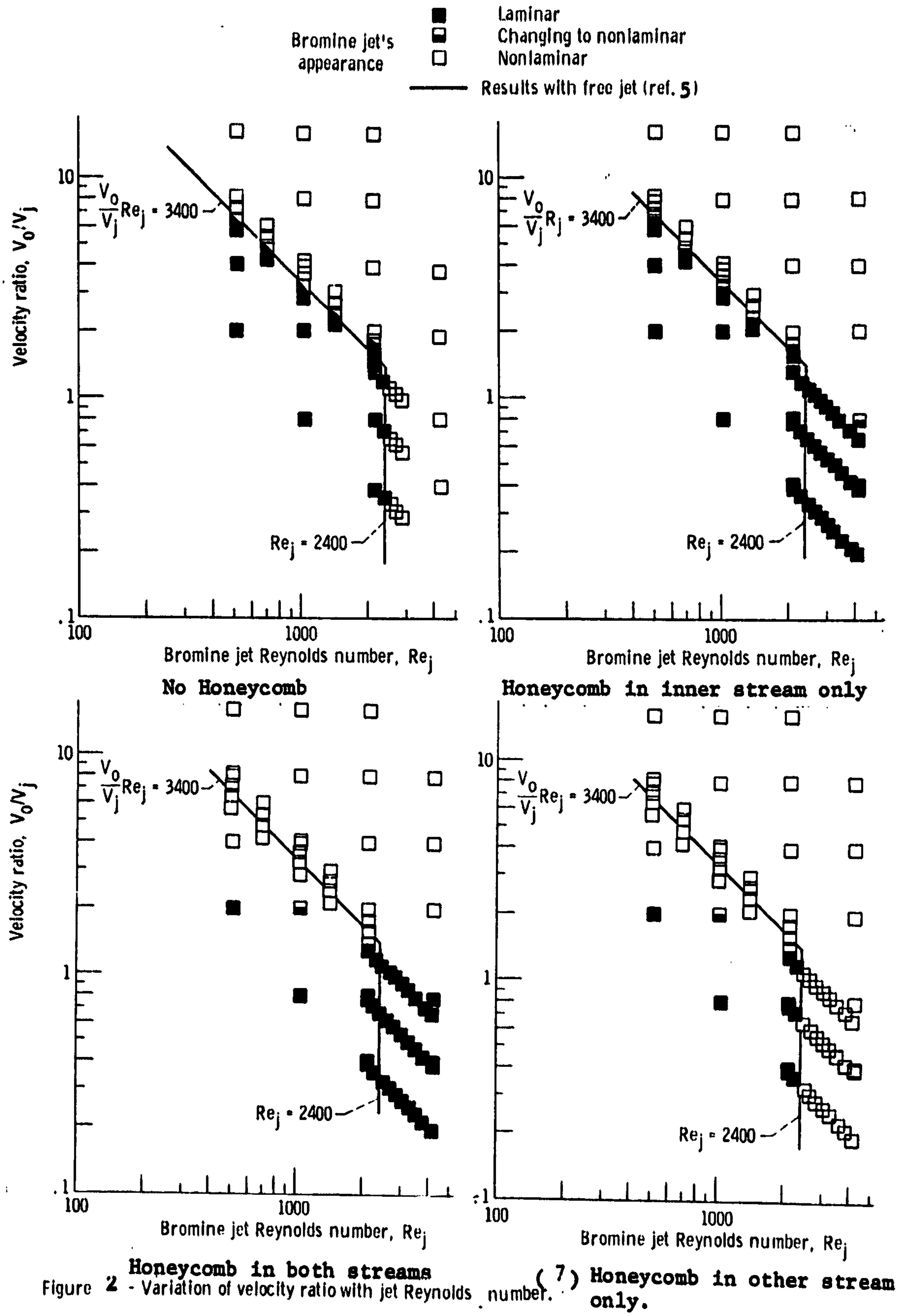


Figure 2 - Variation of velocity ratio with jet Reynolds number. (7)

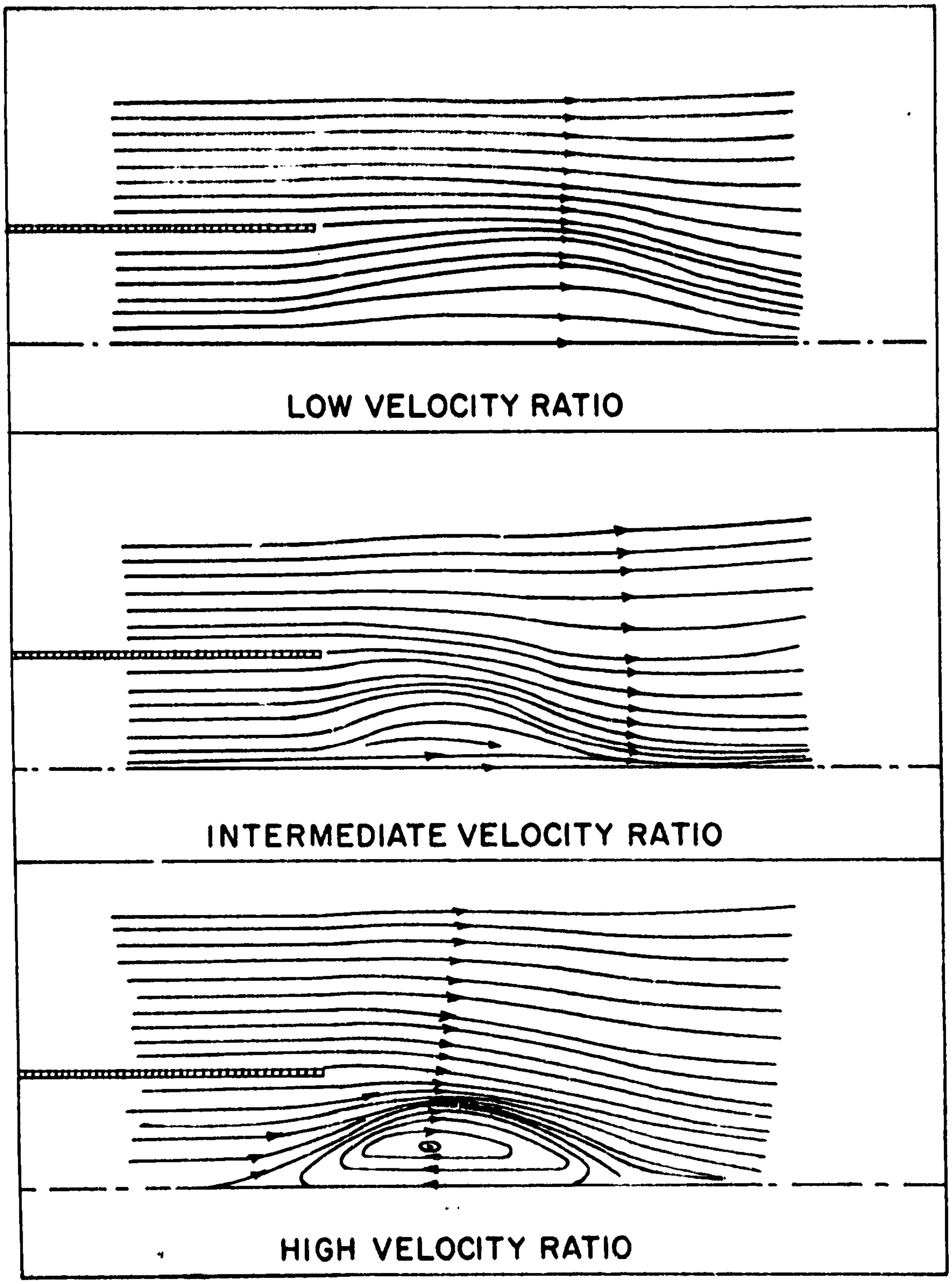
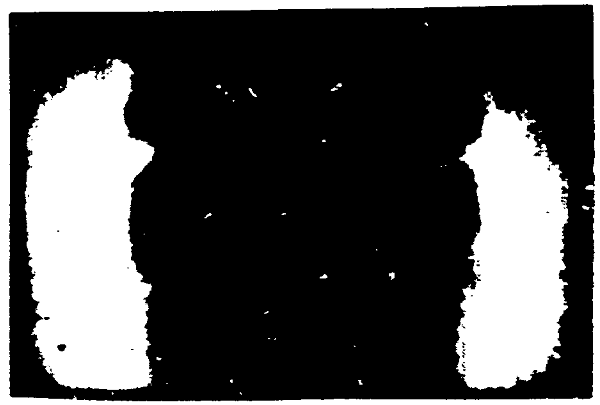
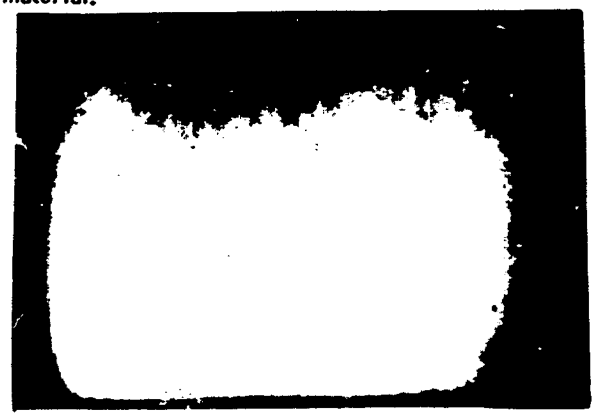


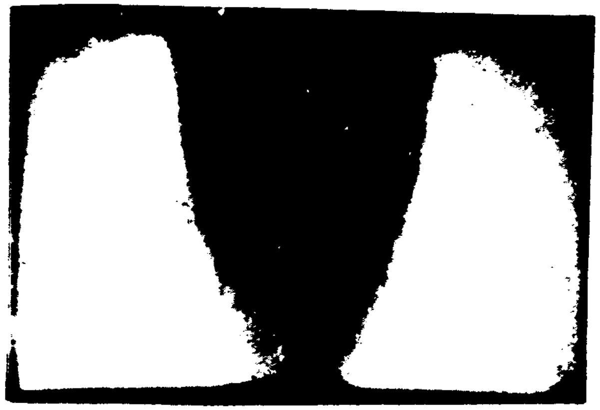
Figure 3 . - Effect of velocity ratio on the stream lines in the initial region of the coaxial jet.



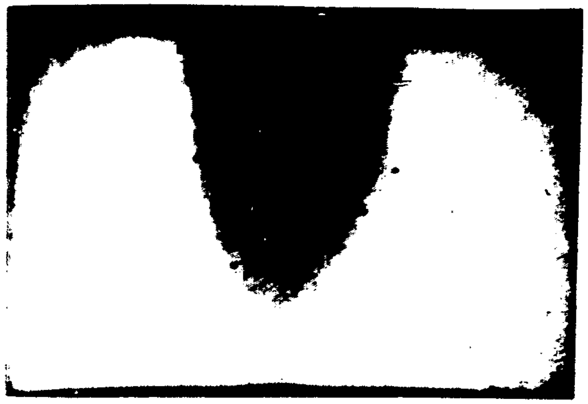
a) - Coaxial flow at a mass flow ratio of 30; no foam material.



b) - Coaxial flow at a mass flow ratio of 50; no foam material.

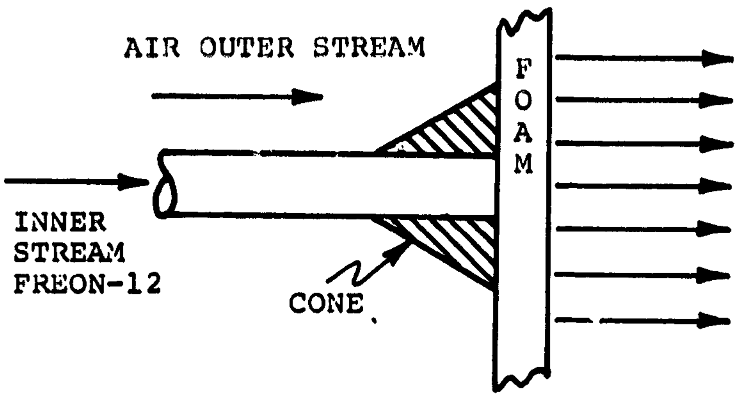


c) - Coaxial flow at a mass flow ratio of 30; with foam material present.

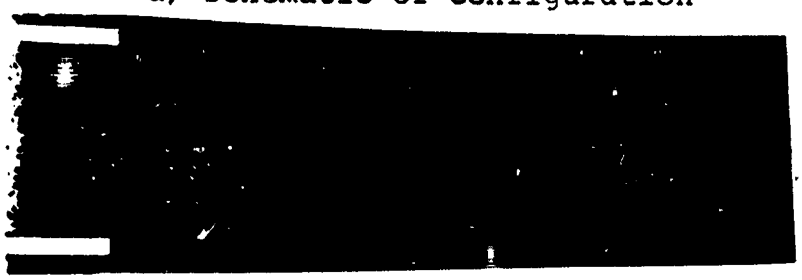


d) - Coaxial flow at a mass flow ratio of 130; with foam material present.

Figure 4. Coaxial Flow With and Without Foam Covered inlet[16]



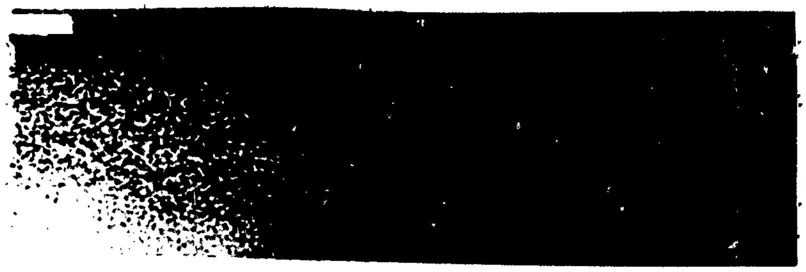
a) Schematic of Configuration



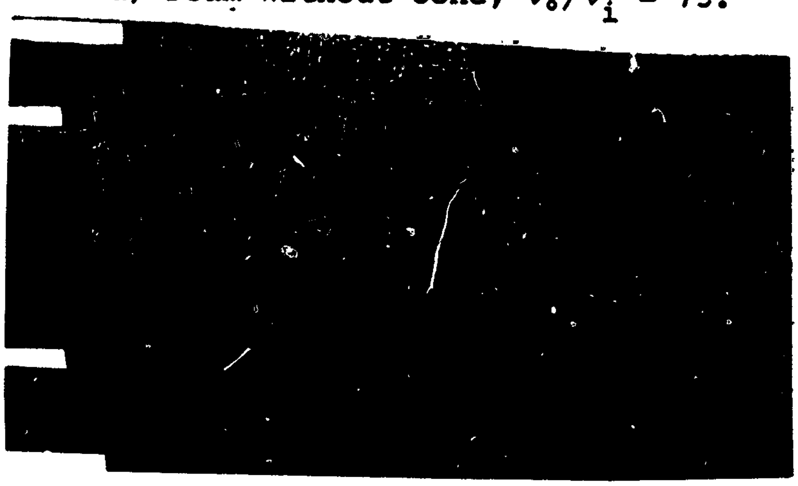
b) Foam Without Cone, $\nu_o/\nu_i = 25.$

c) Foam With Cone, $\nu_o/\nu_i = 25.$

Figure 5. Shadow graphs of a Freon-12-Air Coaxial Jet System Downstream of a Foam Covered Inlet.[18] (continued)



d) Foam Without Cone, $\nu_o/\nu_i = 75$.



e) Foam With Cone, $\nu_o/\nu_i = 75$.

Figure 5. Shadow graphs of a Freon 12 Air Coaxial Jet System Downstream of a Foam Covered Inlet.[18] (concluded)

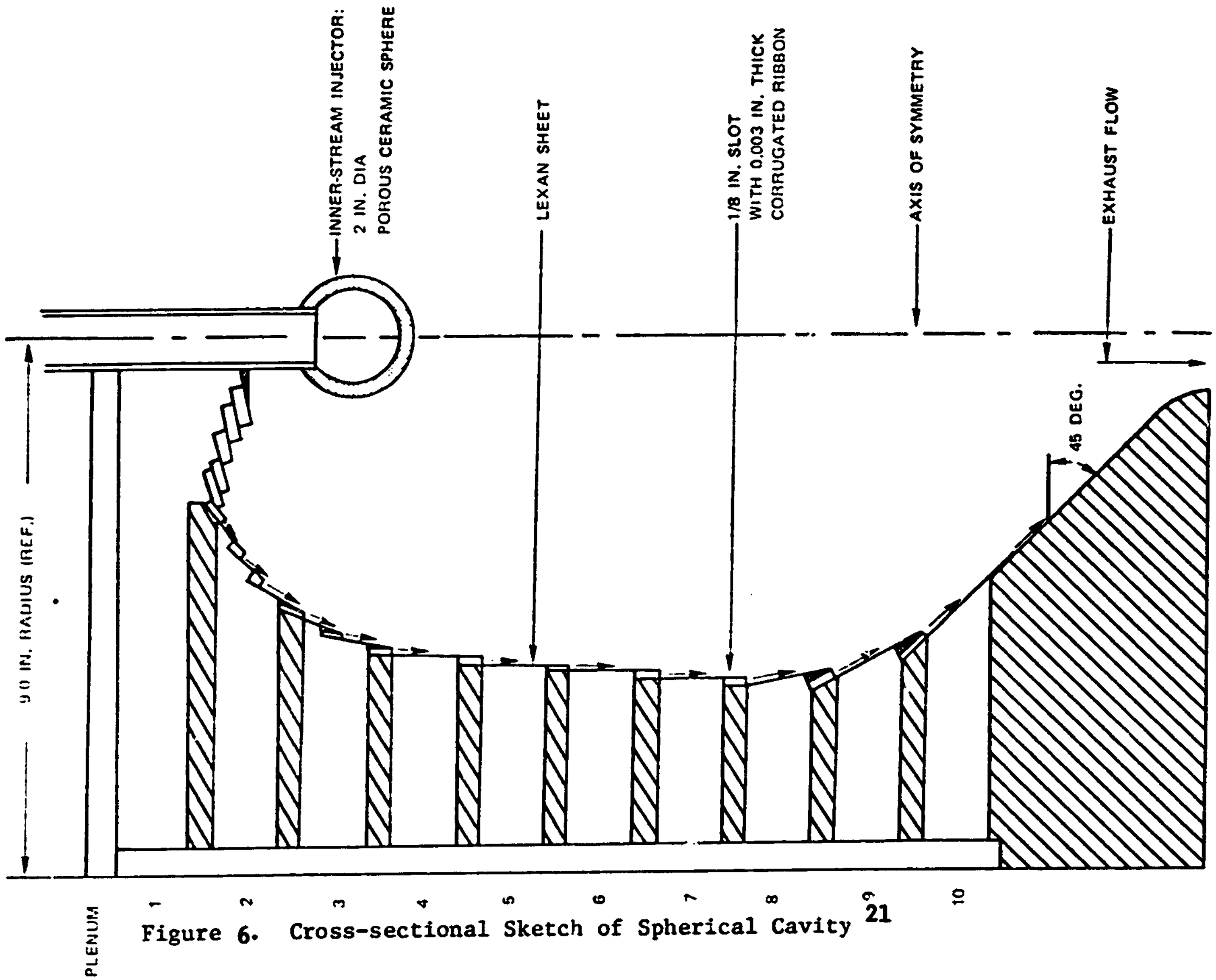


Figure 6. Cross-sectional Sketch of Spherical Cavity [21]

SCHEMATIC OF TEST APPARATUS[17]

SEE FIG. 8 FOR DETAILS OF INLET CONFIGURATIONS

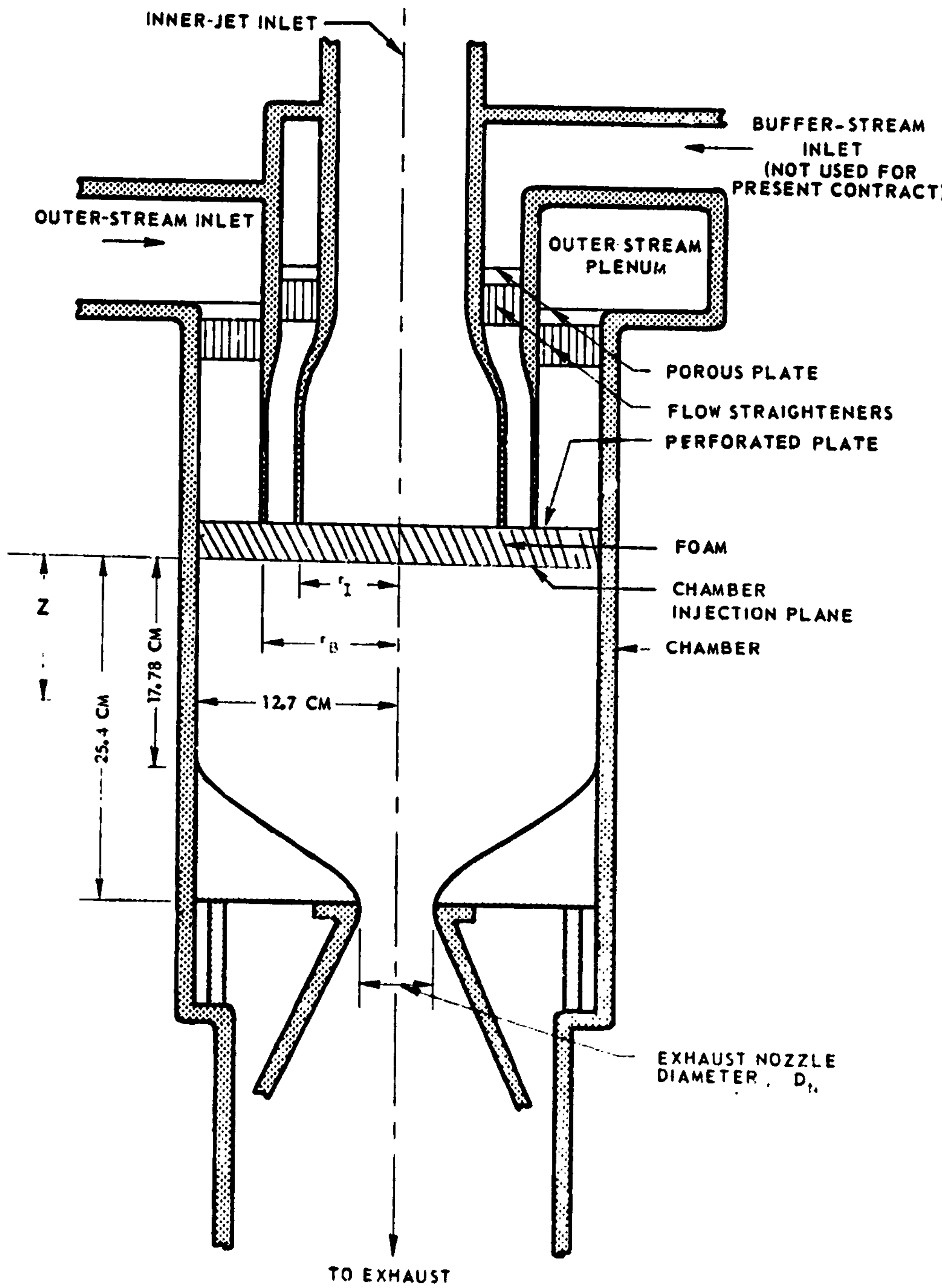


Figure 7.

SKETCHES OF INLET CONFIGURATIONS [17]

d) INNER-JET MANIFOLD RESTRICTED – r_I/r_O 0.60

FOAM CHARACTERISTICS		
INLET NO.	t	N
10	0.953	30
12	0.953	20

INLET NO. 10 USED FOR FINAL DATA RUNS

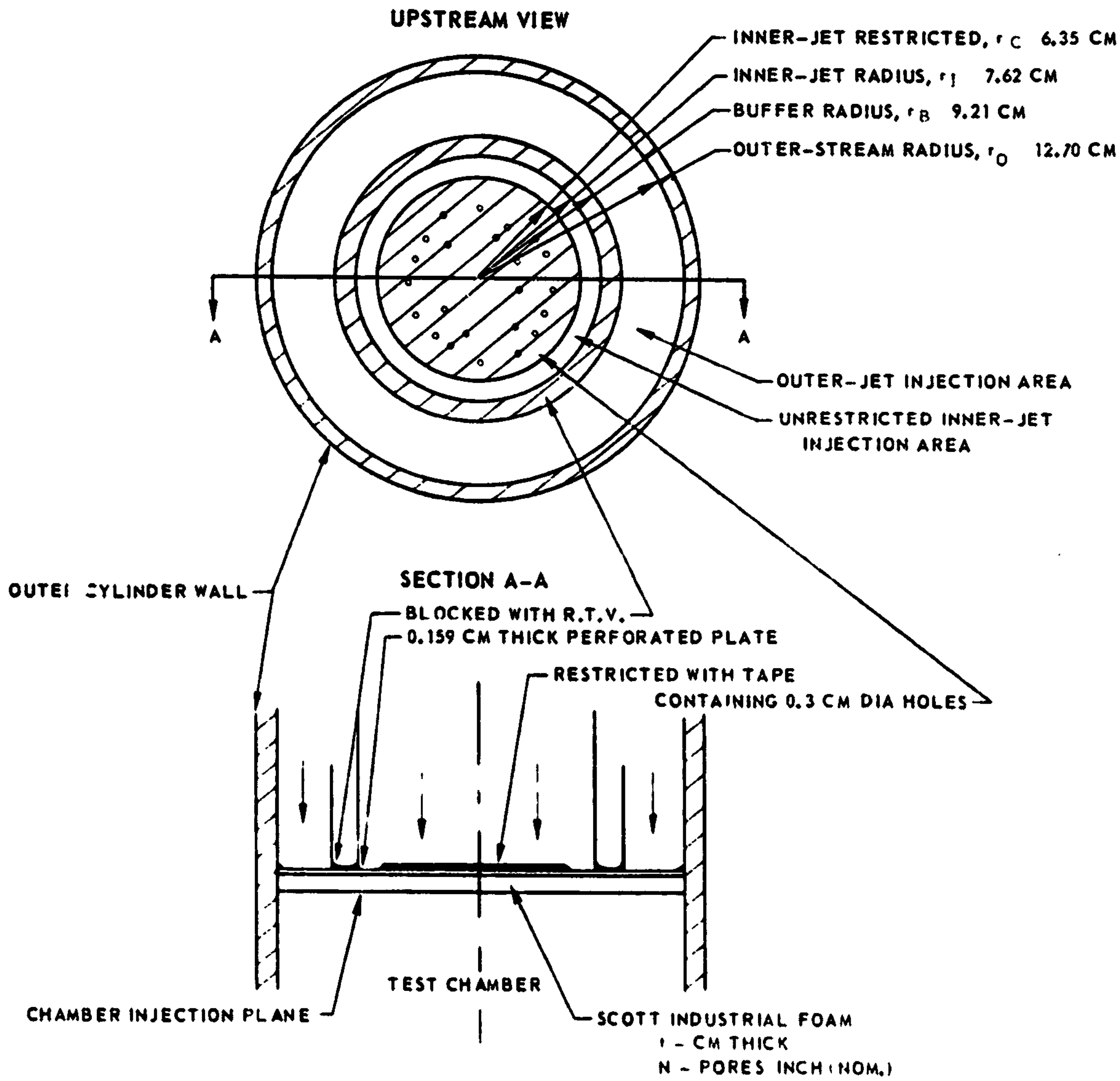


Figure 8.

AXIAL VARIATION OF MEAN VELOCITY PROFILES AT MAXIMUM WEIGHT – FLOW RATIO WITHOUT RECIRCULATION FOR INLET NO. 10

$r_I/r_O = 0.60, \ W_O/W_I = 340$

Z (CM)	RUN NO.	SYMBOL	W(CALC)/W(METER)
2.54	214	○	1.02
7.62	211	□	0.87
15.24	215	△	0.99

$$U_O = \frac{2}{r_O^2}\int_0^{r_O} Ur\,dr$$

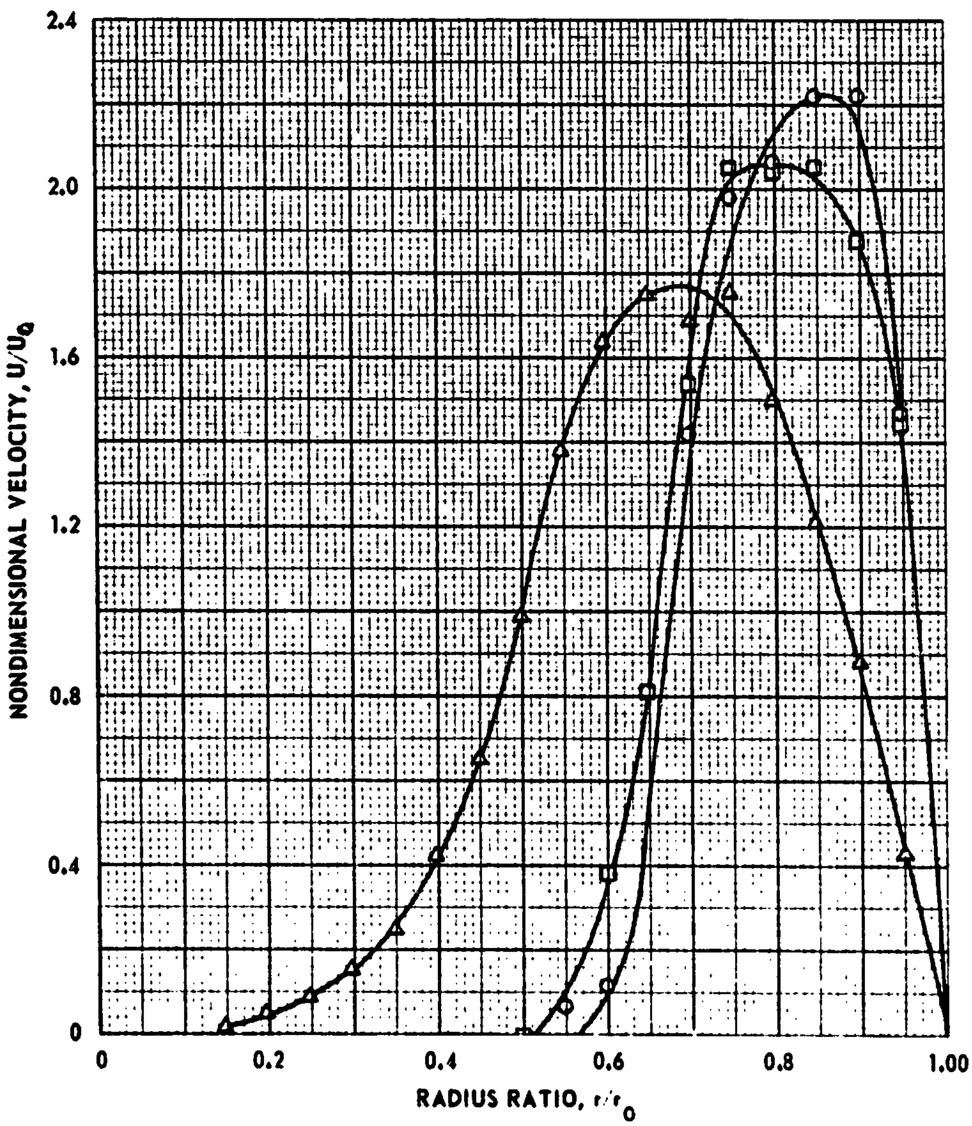


Figure 9.

COMPARISON OF PRESENT AND PREVIOUS CONTAINMENT RESULTS WITH GNR OPEN-CYCLE FLUID MECHANICS REQUIREMENTS[17]

INNER-JET GAS CONCENTRATION DATA OBTAINED WITH CHORDAL ABSORPTOMETER

SYMBOL	INNER-JET GAS	ρ_I/ρ_O	r_I/r_O	SOURCE
○——○	AIR	1.0	0.6 AND 0.7	PRESENT WORK
●——●	FREON-11	4.7	0.7	PRESENT WORK
□----□	AIR	1.0	0.5 AND 0.7	REF. 15
■----■	FREON-11	4.7	0.5	REF. 15

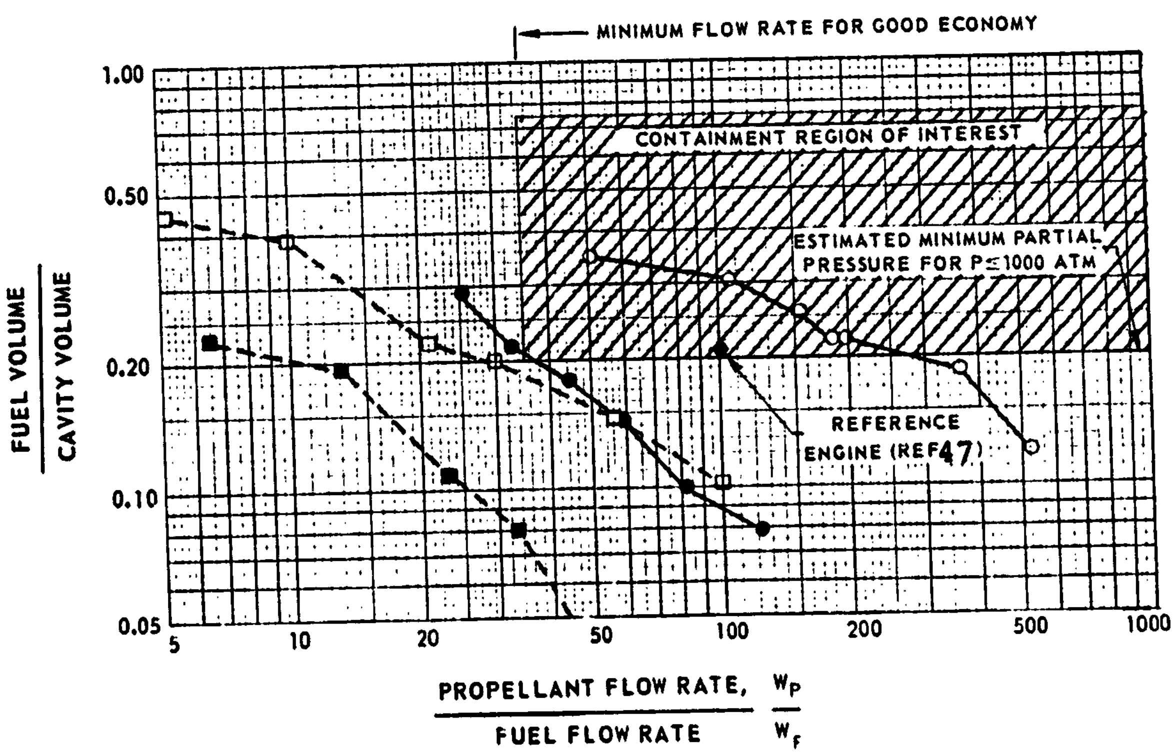


Figure 10.

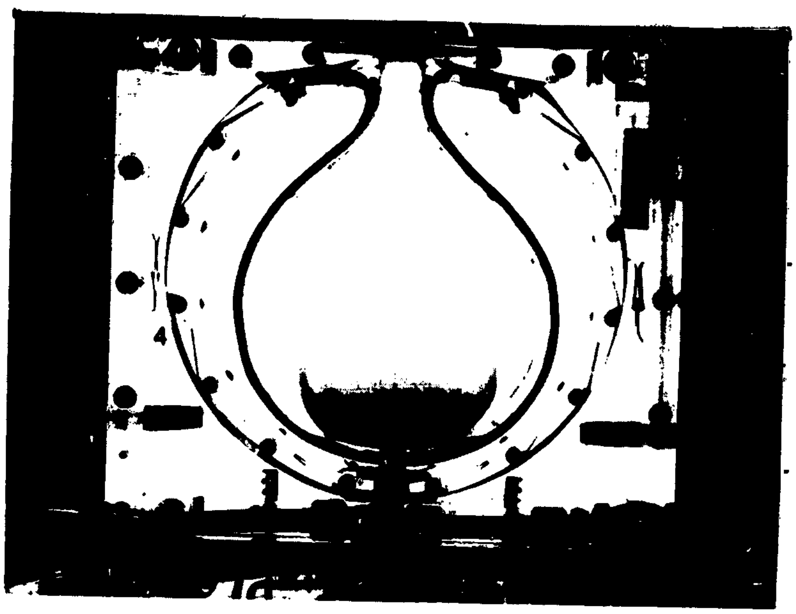


Figure 11.-Flow pattern at a mass flow ratio of 100.[19]

COMPARISON OF PRESENT SPHERICAL CAVITY AND PREVIOUS COAXIAL FLOW CONTAINMENT RESULTS WITH GNR OPEN-CYCLE FLUID MECHANICS REQUIREMENTS[21]

BEST CONTAINMENT RESULTS FROM EACH SOURCE USED IN COMPARISON

SYMBOL	INNER GAS	$\bar{P}_i/P_0$	CAVITY SHAPE	SOURCE	MEASUREMENT TECHNIQUES
△- - - -△	AIR	1.0	'SPHERICAL' – COMBINATION WALL JET	PRESENT WORK	MASS SPECTROMETER
▲- - - -▲	FREON–11	4.7			
○——○	AIR	1.0	CYLINDRICAL – FOAM INLET COAXIAL FLOW	REF.17	CHORDAL LIGHT ABSORPTOMETER
●——●	FREON–11	4.7			
□– – –□	AIR	1.0	CYLINDRICAL – SCREEN INLET COAXIAL FLOW	REF 15	
■–·–·■	FREON–11	4.7			

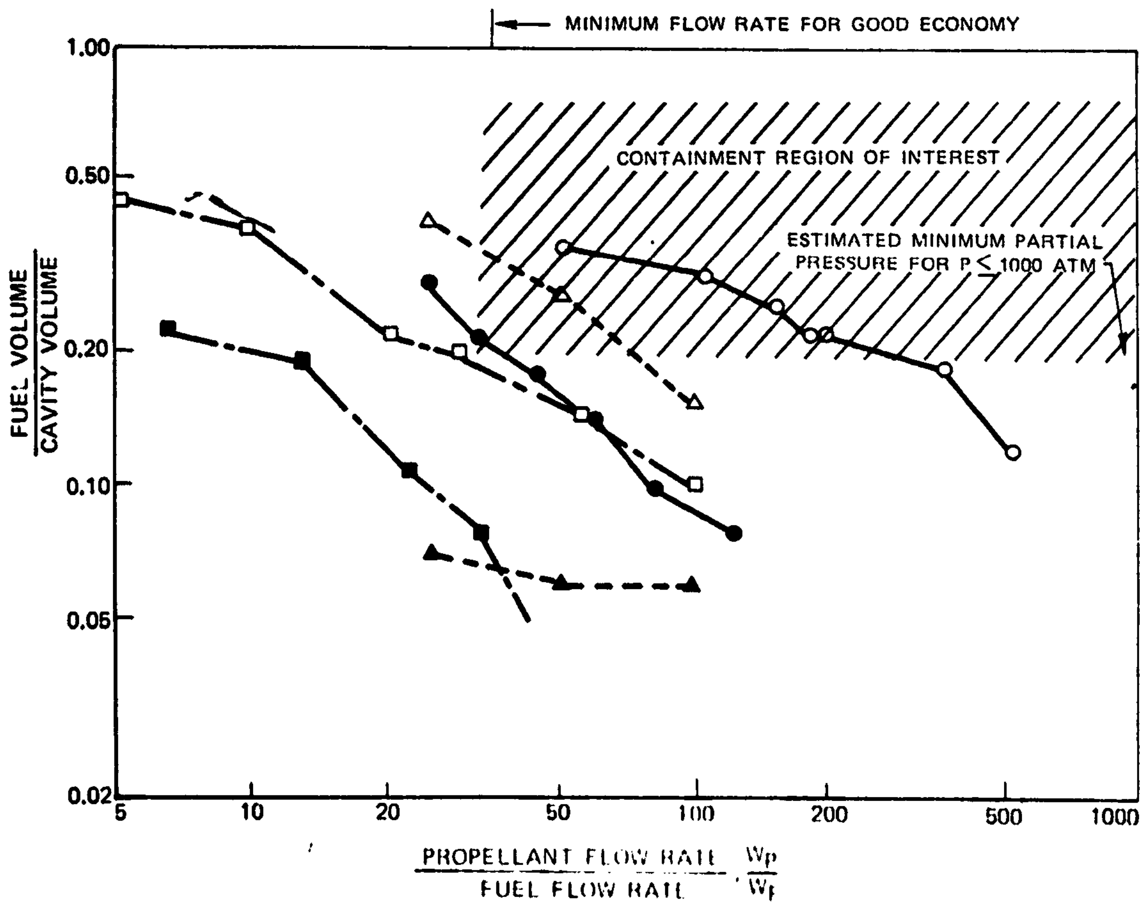


FIG. 12

1 1/2"

2"

2 1/2"

Aerojet Nuclear 2-Dimensional Spherical Coaxial Flow Test Section [24]
Fig. 13 (Injector position effects) 150 cfm outer, 2 cfm inner

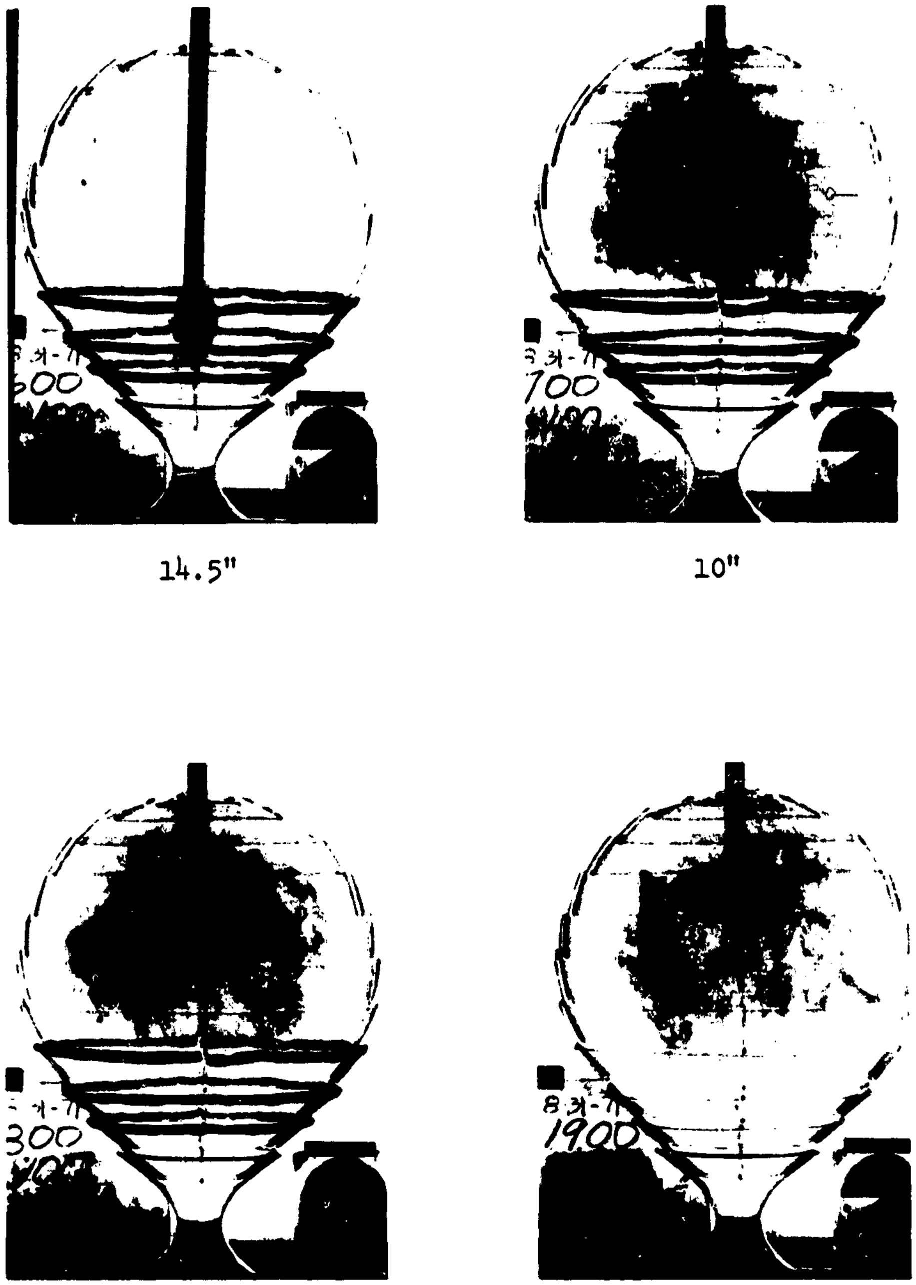


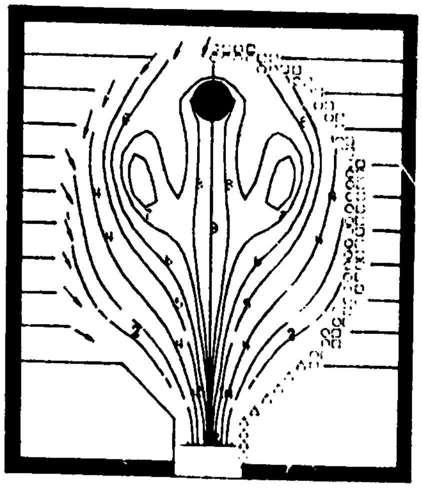
Figure 14. Aerojet Nuclear 3-Dimensional Spherical Coaxial Flow Test Section(24) (Injector position effects) 100 cfm outer, 1 cfm inner.

COMPUTED EFFECT OF DENSITY RATIO ON STREAMLINE AND CONCENTRATION DISTRIBUTIONS [21]

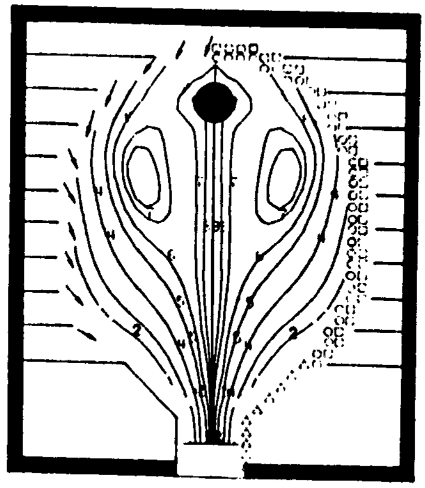
LINE NUMBER	1	2	3	4	5	6	7	8	9	10	11	12
P_i/P	0.90	0.70	0.50	0.30	0.20	0.10	0.05	0.02	0.01	0.005	0.002	0.001
PERCENT OF TOTAL FLOW	100	75	50	25	10	5	2	1	0			

STREAMLINES

STANDARD CONDITION
$\rho_i/\rho_o = 4.7$

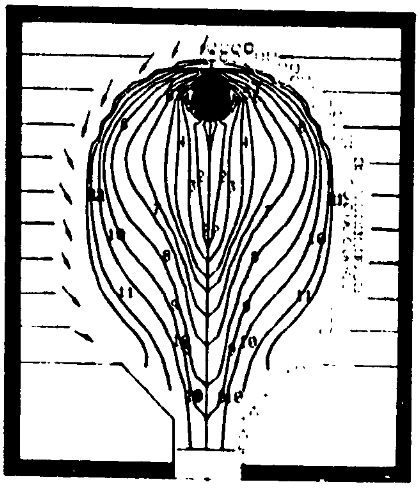


$\rho_i/\rho_o = 1.0$



CONCENTRATION PROFILES

$\rho_i/\rho_o = 4.7$
$\bar{P}_i/P = 0.040$



$\rho_i/\rho_o = 1.0$
$\bar{P}_i/P = 0.053$

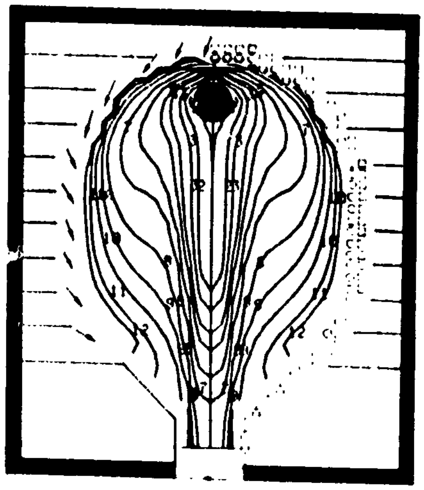


FIGURE 15.

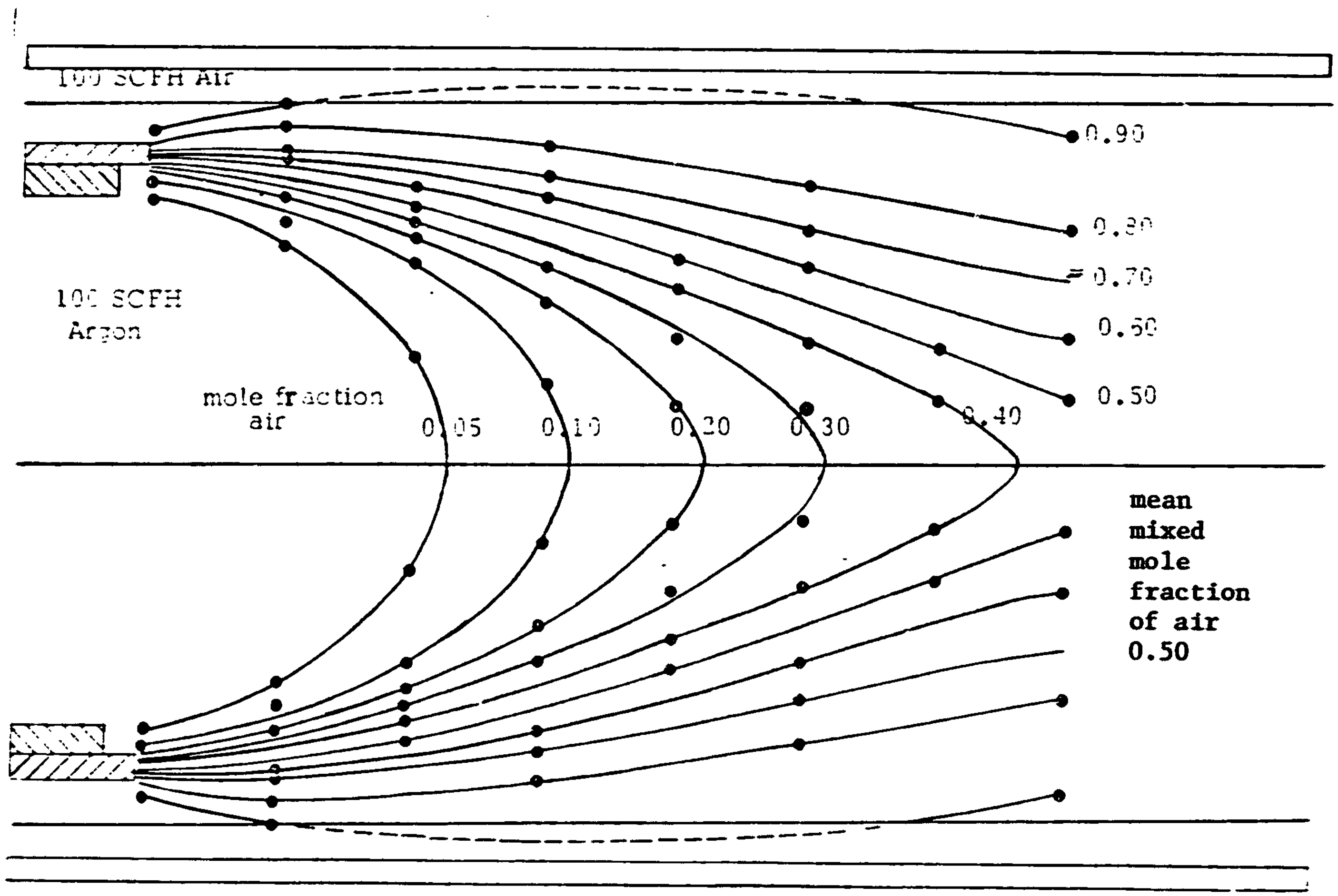


Figure 16. LAMINAR MIXING PATTERN FOR ISOTHERMAL CONDITIONS[40]

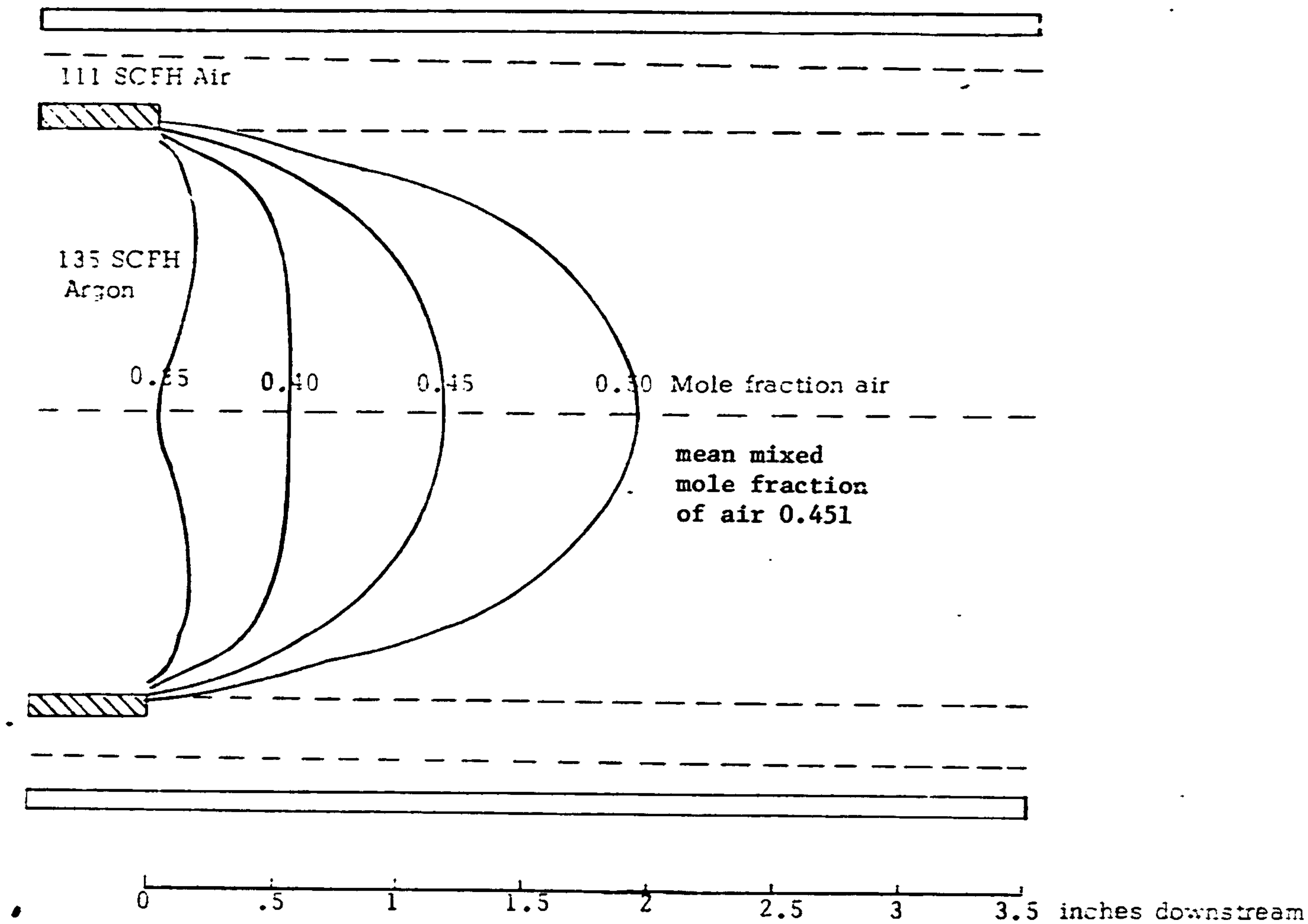


Figure 17. MIXING MAP WITHOUT PLASMA[40]
AIR/ARGON MASS RATIO=0.67

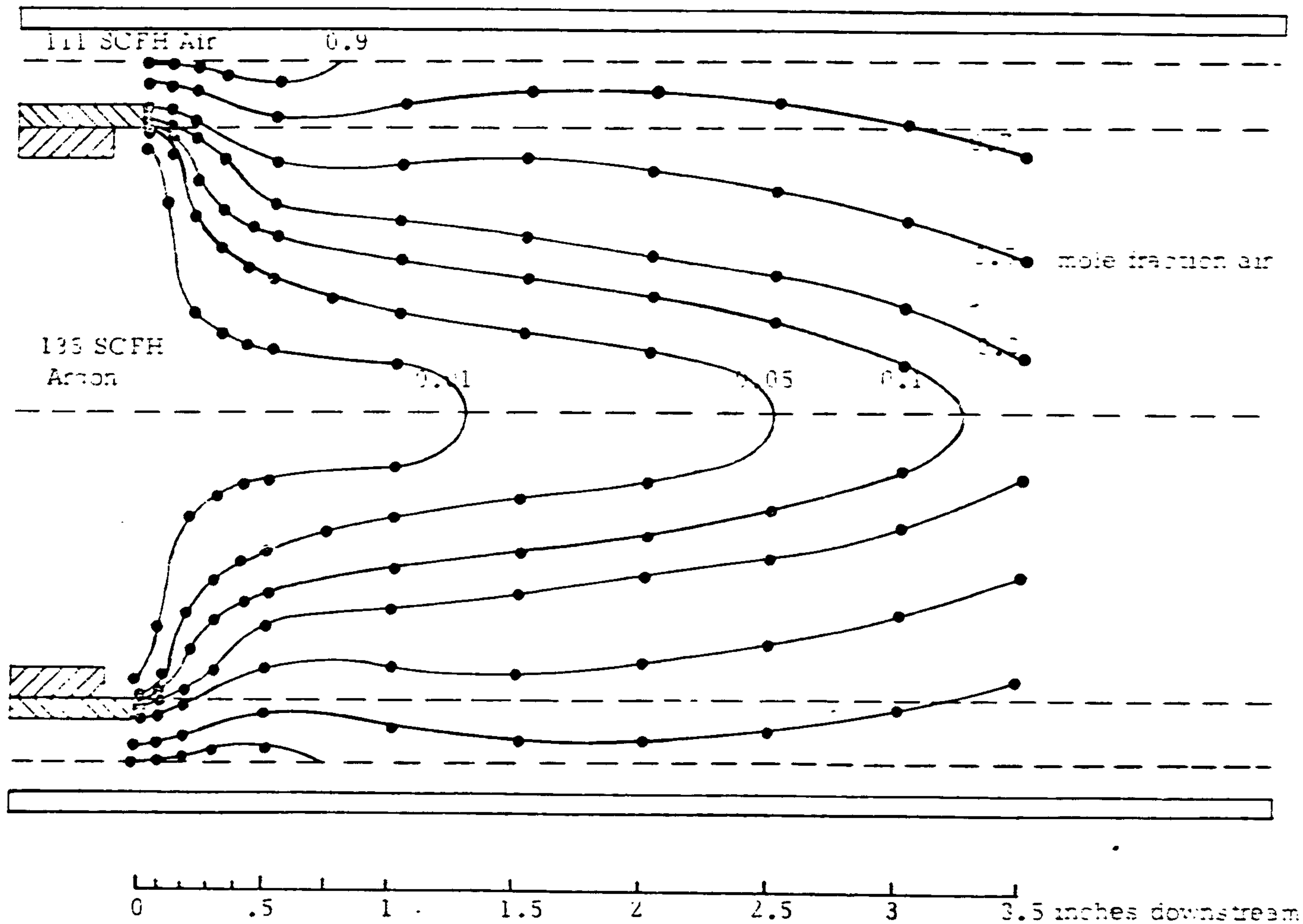


Figure 18. MIXING MAP WITH PLASMA 40
AIR/ARGON MASS RATIO=0.57

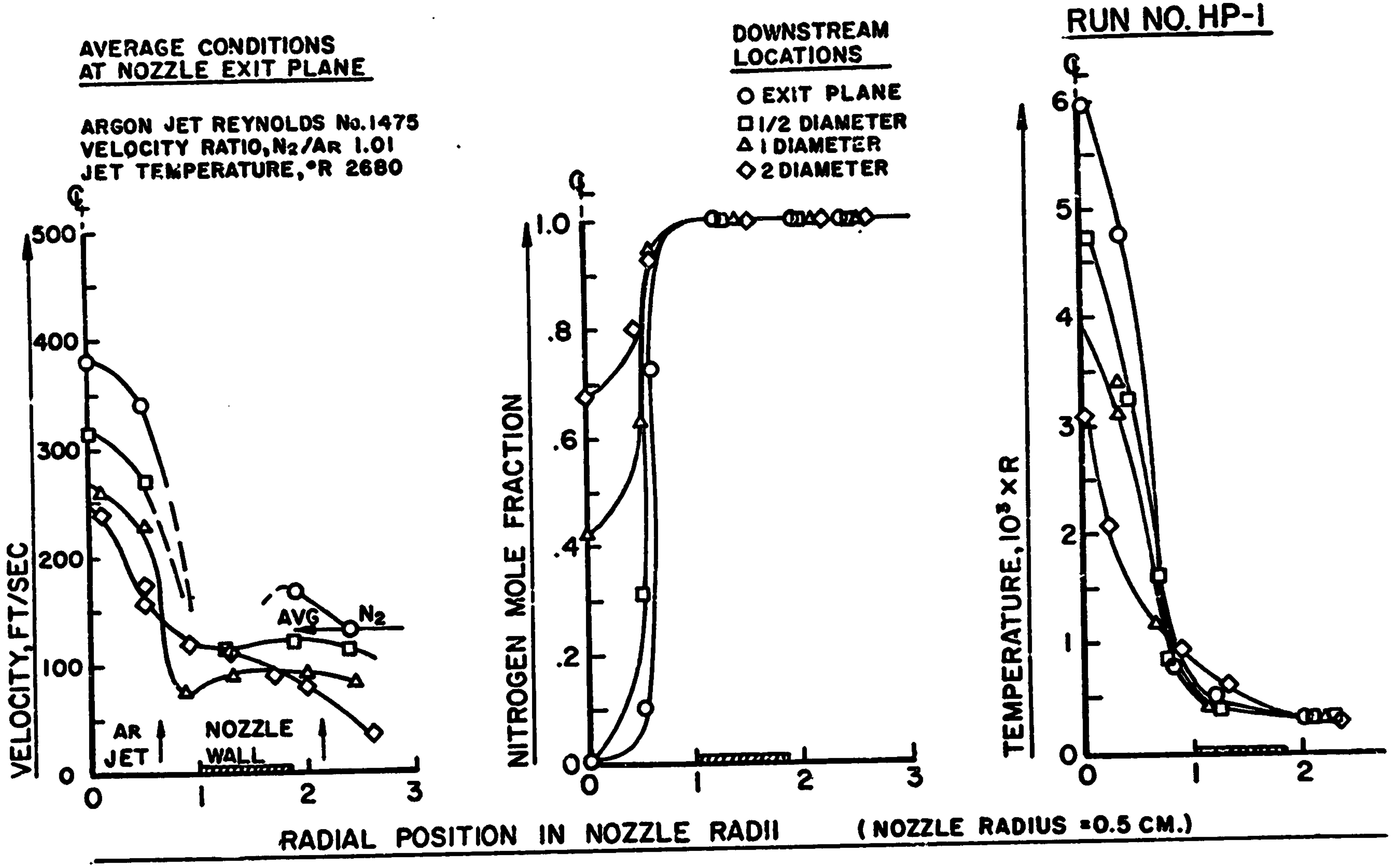


Figure 19.

HALF-PROFILES OF ARGON ARCJET JET MIXING WITH COAXIAL NITROGEN FLOW[23]

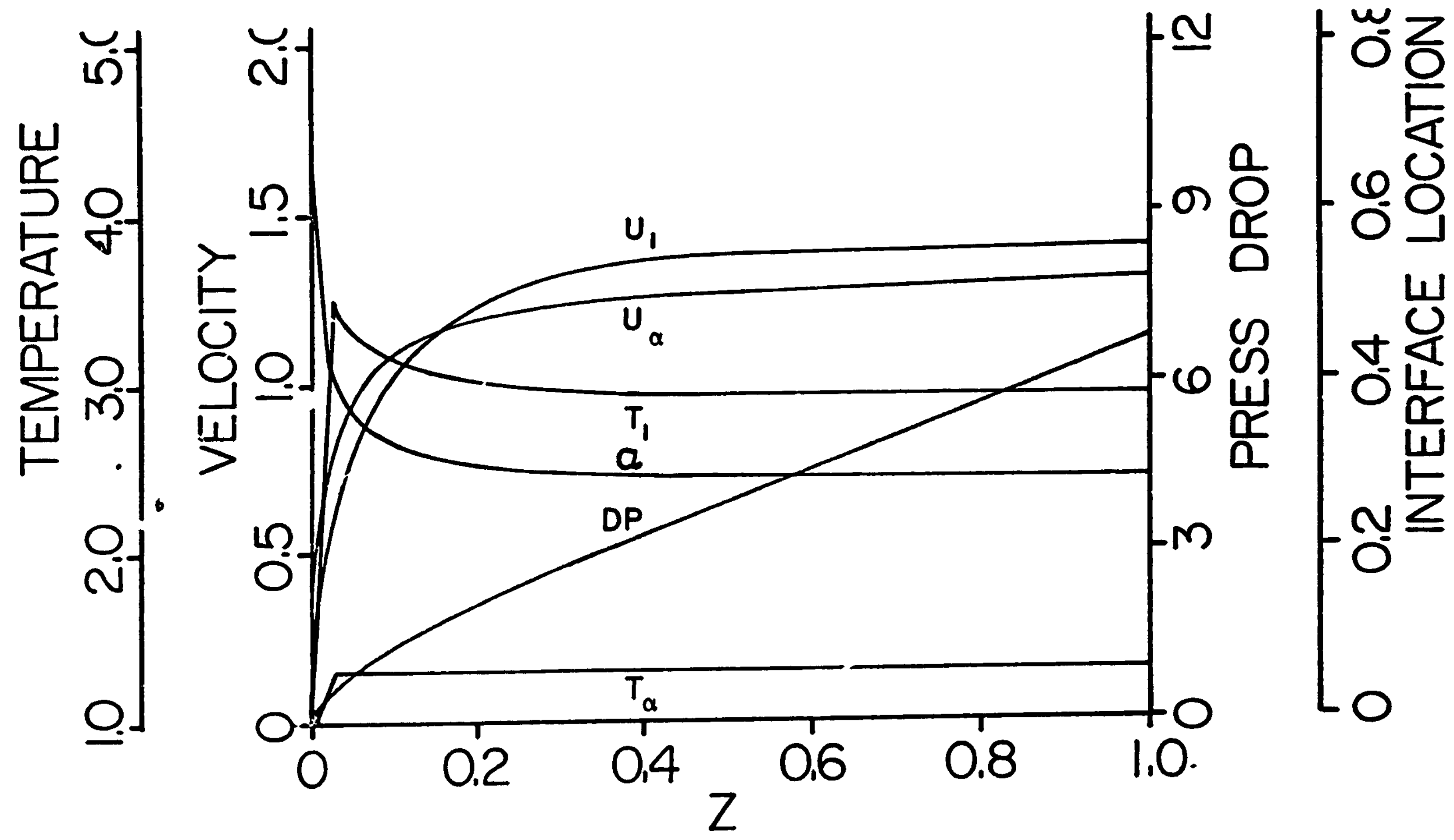


Figure 20. Axial Development of Central and Interfacial Axial Velocities and Temperatures, Pressure Drop and Location of Interface - Constant Generation per Unit Volume Case[45]