Technical Publication

Computer and Physical Flow Modeling in Corner Fired Furnaces

by

Kenneth R. Hules
Senior Staff Engineer
RILEY POWER INC.
a Babcock Power Inc. company
(formerly Riley Stoker Corporation)

John J. Marshall
Manager Program Engineering
RILEY POWER INC.
a Babcock Power Inc. company
(formerly Riley Stoker Corporation)

Perumal Thamaraichelvan
Research Engineer
RILEY POWER INC.
a Babcock Power Inc. company
(formerly Riley Stoker Corporation)

RPI-0115

Presented At

7th Topic Oriented Technical Meeting-TOTeM 7

SCALING AND MATHEMATICAL
MODELING OF COMBUSTION SYSTEMS

27th, 28th and 29th April 1993
Palmer House Hilton Hotel
Chicago, Illinois, USA

Presented jointly by the

International Flame Research Foundation - IFRF
and the
Gas Research Institute - GRI
in co-operation with the
American Flame Research Committee - AFRC

COMPUTER AND PHYSICAL FLOW MODELING IN CORNER FIRED FURNACES

ABSTRACT

Three different techniques of evaluating furnace performance are described with specific application to corner-fired furnaces. In the first section a furnace radiation zone methodology called FASTFIRE, yields results for a vertical bulk gas temperature profile in a pulverized coal corner-fired furnace. A comparison indicates the effect of external staging on the furnace exit gas temperature. In the second section various adaptations of a computational fluid dynamic code describe several ways to apply this engineering resource to practical detail evaluations of furnace gas mixing and flow patterns. Results for an isothermal simulation at elevated temperature and two different simple combustion model simulations indicate velocity vectors at numerous elevations in several different corner-fired furnace configurations. In the third section, physical flow modeling techniques of flow visualization and detailed three dimensional velocity probing provide graphic information on the dynamic flow patterns in corner-fired furnaces with different methods of both internal and external staging of combustion air. All of these techniques yield improved opportunities for successful design in a cost effective and proactive environment to provide quality, performance and reliability in new and retrofit furnace combustion systems.

INTRODUCTION

This paper describes the various furnace modeling techniques currently in use and being developed by Riley Stoker as they apply specifically to corner-fired furnaces. There are three separate sections: 1) Zone methods for radiation heat transfer in corner-fired furnaces; 2) Computational fluid dynamic to detail flow, temperature and chemistry fields in corner-fired furnaces and 3) Three dimensional cold flow modeling in corner-fired furnaces.

SECTION 1. ZONE METHODS FOR RADIATION HEAT TRANSFER IN CORNER-FIRED FURNACES

If furnace flow field details are not needed, simple but powerful zone methods may be used to obtain results for furnace radiation heat transfer, bulk gas temperatures, and Furnace Exit Gas Temperature (FEGT), valid for engineering use. Zone methods apply to many different combustion chamber geometries including wall-fired and corner-fired furnaces. The temperature profile results provide guidance in selecting elevations in the furnace for external staging ports and their effect on such important quantities as FEGT and wall heat fluxes.

Riley has developed a computer program called FASTFIRE for use by engineers in designing or analyzing a furnace. The name FASTFIRE stands for Furnace Analytical Simulation Technique For Infrared Radiation Exchange. FASTFIRE combines zone and Monte Carlo methods to predict overall heat transfer in furnaces.

The original form of FASTFIRE developed by C.E. McHale and R.A. Lisauskas [1] used a "2-dimensional" zone system: a 1-dimensional stack of central core zones going up the furnace surrounded by a stack of ring-like outer zones. This arrangement allowed crude control of mass flow within the furnace (recirculation regions, etc.) and gave a core and near-wall gas temperature at every vertical zone. However, the program was difficult to run. It consisted of several separate modules with overlapping input requirements. More importantly, the need to specify the flow field fairly well in order to obtain good results made it too complex for everyday use in an engineering rather than research setting.

In its current form, FASTFIRE is a single, interactive program based on a 1-dimensional flowfield model with 3-D components in its structure, e.g. separate heat flux calculations for each wall in a zone. The ray-tracing algorithms have been improved considerably to allow use in the interactive, on-line format. The geometry was simplified to a 1-dimensional form for ease of use by engineers in quick turn-around design studies and analyses as in a "what if" approach popular in spreadsheets. One outcome of this simplification is the loss of the core and near-wall gas temperature results; now only a single bulk gas temperature is calculated at each vertical zone. Presently, Riley Research is active in combining the speed and ease of use of the current FASTFIRE with the more sophisticated flow and gas temperature field description of the original FASTFIRE.

Figure 1.1 shows typical results from a FASTFIRE calculation for a corner-fired furnace before and after external staging addition. Normally total combustion air is constant between the two situations. The resultant burner region stoichiometry with external staging leads to higher temperatures in the lower furnace. The lower mass flow with combustion of essentially the same fuel mass yields greater specific enthalpy. The delayed addition of the remaining combustion air in the external staging region then depresses upper furnace temperatures slightly compared to the base case although there is no significant impact on FEGT.

FASTFIRE provides the engineer with a cost effective and efficient tool to evaluate design options for new furnace geometries and in furnace combustion modifications for retrofit systems in wall-fired, corner-fired and other furnace configurations. With the algorithms improved speed, the software runs on personal computers. The run time varies with the specific hardware: for example a 12 MHz 286 system completes completely different furnace geometries in a day while a 50 MHz 486 system takes 30 minutes for a complete design. The major portion of time passes while calculating the exchange coefficients for the geometry. After completing these, the calculation of gas temperatures and heat fluxes as a function of load or different external staging amounts take from ten seconds to one minute depending on the hardware.

SECTION 2. COMPUTATIONAL FLOW DYNAMICS (CFD) TO DETAIL FLOW, TEMPERATURE, AND CHEMISTRY FIELDS IN CORNER-FIRED FURNACES

Computational flow modeling at Riley Stoker provides support for the design and engineering of furnaces, boilers, and combustion systems. In the case of corner and tangential-fired boilers, Riley uses computational flow modeling to evaluate furnace flow patterns under low-NO_x firing conditions and more specifically to achieve the following objectives:

Clarify details of windbox zone bulk flow and local wall flows since they affect low-NO_x combustion strategies, such as the placement and performance of external staging systems;

Parametrically evaluate external staging injection port design and location to find technically sound and cost-effective alternatives;

Evaluate the resultant upper furnace flow pattern and performance with respect to mixing, final burnout, and elimination of locally reducing atmospheres along the waterwalls.

Generally, computational fluid dynamic furnace simulations produce similar results to zone methods or physical flow modeling but with almost infinite amounts of flowfield detail. For these more detailed simulations of wall-fired and corner-fired furnace flow, temperature, and chemistry fields, Riley Stoker uses the commercially-available CFD code FLUENT. FLUENT has the following general features:

Finite volume method for solution of the conservation laws, i.e. equations for mass, momentum (Navier-Stokes equation), energy, and chemical species;

Structured mesh system in 2-D or 3-D using either non-orthogonal body-fitted coordinates or straight Cartesian or polar coordinates;

Model size (number of grids or nodes) limited only by computer power;

Geometry modeling and mesh generation tools;

Steady-state or time-dependent flows;

Incompressible or compressible flows;

Laminar and/or turbulent flows;

Four turbulence models: Re-Normalization Group Theory enhanced "k-ϵ" model, standard "k-ϵ" model, Algebraic Stress Model, and Reynolds Stress Model;

Two-phase chemically reacting and interacting flows (dispersed second phase: solid particles in gases, droplets in gases, bubbles in liquids);

Almost unlimited number of chemical reactions (with independent reaction rates) in the first phase and between phases;

Six-flux radiation model;

Porous media and non-Newtonian media models;

Wide variety of flow, heat transfer, and chemical boundary conditions for surfaces, flow inlets, and global arrangement (i.e. planes of symmetry or cyclic repetition);

The power of a general purpose CFD code with the flexibility of customization for specific applications;

Customization of first phase, second phase, and combustion properties;

Interactive and hardcopy graphics;

Multiple methods of viewing all output quantities for design review & reporting.

During a decade of use, Riley Stoker modified FLUENT in the area of graphics to enhance its use in the design and analysis of furnaces and boilers. However, our present computer power and workstation hardware limitations continue being surpassed and upgraded. Since some of FLUENT's features require use of extra computer power or advanced workstation hardware, Riley Stoker continues developing a consistent method of applying the simpler versions of FLUENT to boiler and corner-fired furnace problems. In particular we use the simpler Cartesian geometry system rather than the non-orthogonal body-fitted system and simplified combustion models.

At the heart of Riley's approach is the issue of compromise: the trade-offs between simulation complexity and detail, available computer power, and time constraints to obtain a solution. In general, when computer speed is limited and closure speedup methods are not yet available, CFD furnace simulations always involve a great deal of compromise. These include the area of grid construction for structured-mesh CFD codes and the choice of models to represent the furnace flow problem physics. Problem solution times are proportional to the number of cells or nodes (geometry) and the number of equations solved (physical models and their complexity) in a problem.

One area where no compromise is possible is the choice between 2-D and 3-D simulations. Current furnace flow problems, especially corner-fired furnace flows, are extremely 3-D in nature, involving interacting main furnace flows with penetration and mixing of burner streams, additional external staged combustion air, and other flow injection jets. The result is a trend toward larger and larger 3-D simulations whose ultimate sizes are regulated by other compromises made in the modeling process.

"Good" grid construction for corner-fired furnace problems involves issues like rate of change of cell size, cell size aspect ratios, cell sizes near walls, cell counts within port boundaries. When limits on these grid features are exceeded, numerical problems and inaccuracies may invalidate the computed results. Furnace flow simulations put immediate pressure on these good grid construction guidelines. For example, furnace inlet ports may be 6 inches in diameter and the furnace itself may exceed 100 feet in size. Such length scale discrepancies invariably lead to large models, i.e. a large number of nodes, which leads directly to long solution times.

In order to control the number of cells, almost all of the grid construction guidelines may be stretched beyond their "good" limits. Rates of change of adjacent cell sizes will go as high as 1.5 or 1.6 instead of the desired maximum 1.2. Cell aspect ratios (between any 2 of the 3 cell dimensions) will go as high as 10:1 instead of the desired maximum 4:1. In almost all cases the minimum number of cells is used to define a port: 2 cells each in orthogonal directions (e.g. height and width or width and depth) for a total of 4 cells inside the port. Fewer cells defining a port gives poor jet penetration, spreading, and mixing results. Usually defining the minimum number of horizontal cells at an external staging port location provides enough cells to set up the horizontal definition of burner zones, although sometimes burner zone horizontal dimensions add to the number of cells needed. Usually one of the last steps is coordinating the vertical cell sizes needed in the hopper and burner zones with the horizontal cell spacings. Finally, a slightly larger cell size is used from the external staging elevation to the furnace roof to minimize total cell count. Any interior waterwall features in the upper furnace (platens, pendants) are accommodated by the best possible grid fixed by the lower furnace geometry. More time is spent determining the good use of nodes than in the actual setup of the problem on the computer.

Even though every corner is cut as fine as possible in the geometry as described above, typical corner-fired furnace models range from 80,000 to 200,000+ cells. These large problems often require careful selection of physical models used in the simulation, especially in turbulence and combustion modeling. The corner-fired furnace flow field is 3-D which means a minimum of 4 equations are being solved: pressure (P) and three velocities along the orthogonal axes (U, V, W). The turbulent flow requires the choice of a turbulence model radically alters the equation count. The standard k-ϵ model minimizes the equation count, now at 6. In corner-fired furnace problems the standard k-ϵ turbulence model is adequate for the swirling main flow and small recirculation regions near the windboxes.

For fastest turn-around, an isothermal model extending only up to the furnace arch may be used since no equations are added for enthalpy or chemical species for combustion. These models are not as useful as those with combustion, but they can give a reasonable view of a base flow condition in a corner-fired furnace when the time constraints are severe. This modeling technique is similar to a physical flow model, however, the computational model uses full scale and can be set at an elevated gas temperature. A 1204.4°C (2200°F) model temperature gives a reasonable bulk viscosity value. Windbox inlet velocities are the same between field and the simulation. A Thring-Newby type of adjustment is made in the windbox width while the vertical dimensions are unchanged to retain the proper elevations of coal and auxiliary air compartments. This correction brings the proper mass flow into the furnace and approximates combustion region momentum flow into the furnace main flow field.

Velocity vectors become the main visualization tool since there are no gradients of temperature or chemical species to indicate mixing or other flow field features. The k-ϵ model parameters, kinetic energy of turbulence and eddy dissipation rate, could be used to indicate mixing but clients usually do not have experience of or a good feel for these quantities.

Figures 2.1 and 2.2 show results typical of this type of corner-fired furnace simulation. The effects of mass addition as a function of height along the windbox are displayed. Also the origin and extent of the almost inactive central core are revealed. The low activity and low velocity central core extends from the furnace floor well up into the upper furnace depending on the arch size and presence of platens or pendants.

For external staging design and a more complete picture of the base furnace performance, temperature effects and a combustion model must be added to the simulation. If a complicated combustion model is used, then the number of equations being solved increases dramatically and solution times soar. For coal firing a simple combustion model burns CO as fuel with O_2 as oxidant to produce CO_2 as product with a higher heating value of 10,111.2 kJ/kg (4347 Btu/lbm). This system adds 4 equations to the count (enthalpy, and fuel, oxidant, and product species) bringing the total to 10. Visualization of results now extended from just velocity vectors to include contours of temperature, CO, O_2, and CO_2 if desired.

The coal input flow (excluding the ash) is converted to an equivalent all gas phase input containing enough CO to produce the same total heat release. The simulation uses field values for gas temperatures and exit velocities in windbox compartments. As a result, simulation inlet densities duplicate the field and requiring no distortion of the compartment free areas. However the O_2 content of the coal compartment streams is adjusted. In the field, coal compartments have primary and secondary air streams along with the coal. In the simulation, coal compartment O_2 content is adjusted by the sum of the O_2 flow needed to burn the CO flow completely plus the free O_2 flow needed at the exit to match the excess air condition minus the O_2 flow available from the auxiliary air compartments. This calculation assumes complete combustion of the coal with no carbon loss in the ash. Usually this calculation produces some O_2 flow in the coal compartment streams for initial CO combustion rather than relying entirely on O_2 mixing from the adjacent auxiliary air compartments. An initial guess of a 1204.4°C (2200°F) uniform furnace temperature provides a mathematical "ignitor" to stimulate combustion of the relatively cool compartment flows, 148.9°C (300°F) to 371.1°C (700°F). The "flame" extent and peak temperature is controlled by adjusting the CO reaction's Arrhenius rate constants to retard the conversion rate. This simple model allows tracking of the important species CO, where all unburned hydrocarbons have become equivalent amounts of CO.

Figures 2.3 and 2.4 show results of this type of simulation for the base case (before external staging) of a corner-fires furnace. The flame regions coalesce into a ring structure showing the origin of "hot" and "cold" corners. The relatively cool and O_2 poor central core is flame free with very low velocity and extends from the hopper exit up to the arch. CO tends to collect in the corners rather than in the central core. Whether or not the central swirl is circular or elliptical depends on the furnace aspect ratio and the placement of windboxes along the walls or in the corners.

When an external staging design is considered, a new combustion model may have to be used. If the excess air condition is not too low and the staging is not too deep, then the O_2 calculation made above for the coal compartment streams gives a positive number, and the CO combustion model may be used. However, in some cases the excess air condition is low and the staging is deep enough that the O_2 calculation gives a negative number. In this case a new combustion model is needed. Riley has been developing two models for this situation. The first is a modification of the CO model detailed above. In this case the oversupply of O_2 is taken from the O_2 in the auxiliary compartments only, the external staging streams carry normal O_2 content thus preserving final burnout region conditions and exit excess air. This model also retains direct observation of the CO for monitoring external staged air mixing and burnout. A disadvantage is that lower furnace free O_2 content and distribution are distorted even more severely.

Another single reaction model burns Carbon (C) with a higher heating value of 33,727.0 kJ/kg (14,500 Btu/lbm) and O_2 as oxidant to produce CO2. Converting the coal to an equivalent gaseous flow of C produces the total heat release. Inlet streams, including the coal compartment, carry their normal O_2 contents. Again, the Arrhenius rate constant adjustment retards combustion to control "flame" peak temperatures. This models advantage allows nearly undistorted furnace O_2 content and distribution. Coal compartment streams carry relatively high O_2 content so combustion begins with less mixing needed from auxiliary air streams. A disadvantages is direct tracking of the CO species is lost and the degree of final burnout may be distorted.

Figures 2.5 and 2.6 show results of this type of combustion model for a corner-fired furnace with an external staging design that exploits the rotational character of the main flow. The overall character of lower furnace flow behavior is similar to the base case. Since the central core is essentially inactive, mixing of the staged air must be restrained to the wall and rotating ring regions. Final burnout and elimination of reducing environments along the walls require distribution of external staging O_2 close to the walls. The figures show the mixing and performance of this external staging application.

Figures 2.7 and 2.8 show the results of a front wall/ rear wall external staging system usually seen in wall-fired furnaces. The unit and flow conditions are the same as above so that the carbon combustion model was used again. In this application of a wall-fired Over Fire Air (OFA) design to a corner-fired unit, the goal is not to over-penetrate the staging air through the main circulation into the core. This would waste the O_2, yield incomplete burnout, and leave a reducing-type atmosphere on the waterwalls. The figures show that with proper port sizing, even the front wall/rear wall OFA design can produce good mixing and burnout. In some respects this design performs better technically than the other four port arrangement. However, this design is less cost-effective since it has more waterwall penetrations.

Computational fluid dynamics proves to be a very effective tool in aiding the engineer in evaluating potential success and impact of advanced alternatives. It greatly enhances the use of available resources to narrow down and eliminate ineffective designs that would prove to be a burden to both the developer and ultimately the user.

SECTION 3 THREE DIMENSIONAL COLD FLOW MODELING IN CORNER-FIRED FURNACES

In boiler systems significant effort occurs in designing the shape and size of components for the proper flow distribution of air, gas and fuel in all parts of the system. The goal is to minimize energy losses from unnecessary pressure drops, poor combustion efficiency, poor heat transfer to furnace walls, slagging, fouling, corrosion, erosion and byproduct combustion emissions.

Without access to supercomputers, advanced computerized flow codes on conventional main frames, predict flow fields in steady state nondynamic situations for the large furnace models detailed above. These codes are cost effective tools, but without supercomputers limit analysis of flow fields containing oscillating flows, vortex shedding and other frequently nonperiodic phenomena. Although manufacturers and others continue to improve the predictability of these tools, the use of scaled physical flow models is often the most cost effective method for evaluating dynamic furnace flow conditions.

Physical model studies are more empirical than rigorous theory and less costly than prototype experiments. They offer quick and cost effective routes to development and design. The objective is to predict conditions inside full size field equipment (the prototype), by running experiments on a model smaller in size and operated at a lower temperature.

THE CATEGORIES OF PHYSICAL MODELS

Physical flow models are categorized by their geometry, working fluid and temperature. Two or three dimensional, they use air, water, acid/alkali or gas at cold, warm or hot temperatures. The type chosen depends on the objectives of the investigation. Two dimensional model define gross flow patterns by indicating areas where detailed three dimensional model may provide more refined information. Frequently water models are two dimensional and cold flow. They provide quick, qualitative visualization in open channels on a water table. Fuel and combustion air jet mixing studies use acid/alkali. Aqueous alkali dosed with an indicator represents the fuel and dilute acid represents the combustion air.

Complete flow patterns are obtained with three dimensional models. Velocity probes provide the velocity profiles at selected locations. Although air, gas or water can be used, cold air is the most common working fluid.

PHYSICAL MODEL SIMILARITY CRITERIA

For a good correspondence between model and prototype results, similarities must exist between them. These similarity criteria are often represented by non-dimensional groups such as the Reynolds, Froude, Grashoff, Prandtl and Schmidt numbers. Some discretion and experienced judgement is needed in selecting the criteria used and how closely they are controlled. The goal remains to achieve as much similarity as possible in the phenomena under investigation to allow for good correspondence of results.

Several types of similarity criteria exist for flow systems. Geometric similarity is simplest. For every linear dimension in the prototype there is a corresponding dimension in the model at a constant reduced ratio or scale. This is best used in isothermal non-compressible flow systems such as duct work. It may be partially abandoned when chemical reaction and non-isothermal processes occur such as in furnaces.

Three categories of mechanical similarity exist: static, kinematic and dynamic. Static similarity requires that the model and the prototype have similar elastic or plastic deformation when a stress is applied. It is not applicable in duct or furnace models. Kinematic similarity means fluid or solid particles follow similar geometric paths in corresponding intervals of time.

Dynamic similarity holds constant force ratios that cause mass acceleration. For dynamic similarity on a water table model, the non-dimensional Reynolds Number (Re), which represents the ratio of fluid inertial and viscous forces, should be the same in both the model and prototype.

The kinematic viscosity of hot furnace gases is about 120 times that of cold water. Hence in a one twelfth scale model, the water velocity need only be one tenth of the prototype. For this reason water table models with reflective powders sprinkled on the surface are very suitable for flow visualization. These models are best used for flow that is essentially parallel in adjacent planes. It readily illustrates flow fluctuations and separations. The major disadvantage is the qualitative limitation of results. Where the flow is not parallel the results are very misleading and error can result in design of the full scale unit.

To obtain the same Re in three dimensional flow models, a one twelfth scale model should be run with the same velocity as the prototype, because the cold air has a viscosity of about one twelfth that of furnace gases. When Re in the model remains above 6,000 at a reduced velocity, turbulence is maintained and representative mixing and flow pattern information can be obtained. To be sure of sufficient turbulence, 10,000 is most often used by modelers as the minimum Re.

The final two types of similarity criteria considered here are thermal and chemical. Neither thermal nor chemical similarity are used in furnace or duct models where flow field phenomena are being evaluated.

Similarity Criteria For Furnace Models

The dynamic processes in a combustion furnace present the modeler with a situation that can not be completely simulated in a physical model. Beyond the six similarity criteria already mentioned, over a hundred others can be shown for the combustion process. Many of these are mutually independent. Similarity in one criteria creates error in another. In practice, partial modelling is used to simulate the dominant effects controlling the phenomena under investigation.

Most often interest lies in the fluid mechanical features of the furnace, such as jet penetration, velocity profiles and other flow patterns. Both geometric and dynamic similarity criteria are used. Sometimes it is necessary to distort certain parts of the geometry, as in the burner jets, to achieve similarities in other parts of the model, for example in the upper furnace. It is also practical to run at velocities less than the prototype while still maintaining a good level of turbulence and dynamic similarity.

Furnace flow field information can be obtained in an isothermal model. Two general flow regions exist in the furnace shown in **Figure 3.1** [2]. This simplification indicates two different flow regions. A small Region I is located from fuel and combustion air jet entry to an effective combined jet flame front. This is the most active region of combustion that includes: fuel and air mixing; fuel heating and de-volatilization; flame stabilization; high temperature and chemical concentration gradients; rapid changes in density; adjacent furnace gas entrainment and the combining of flames from different burners.

The much larger Region II is considered relatively isothermal. Scaling to an acceptable degree of accuracy is possible but depends on getting the geometry and end conditions correct.

The jet issuing from a single burner into a large open furnace can be considered a free jet. **Figure 3.2** shows a typical single burner free jet under combustion conditions. Flow from a free jet burner will expand and entrain gas up to the combustion front. At the combustion front there is a rapid decrease in density and increase in cross sectional area which greatly affects the momentum flux. **Figure 3.3** shows an isothermal free jet superimposed on the combustion free jet. It can be seen that the isothermal jet is not similar to the combustion jet due to the lack of combustion.

The same is true for large multiburner systems. Here the problem is compounded by the interaction of adjacent burner nozzles. Systems with the ratio of burner nozzle spacing to the burner nozzle diameter (L/D) greater than 18 operate as independent nozzles as shown in **Figure 3.4** [3]. Multi burner systems with L/D less than 18 can be treated as single burner. **Figure 3.5** is typical of most utility furnaces where the L/D is approximately 3. Therefore, the multi nozzle assembly of the corner-fired furnace windbox can be treated as a single burner assembly.

A long standing method of accounting for the area change in a free jet makes the burner opening at the wall larger. This method is known as Thring-Newby port distortion. The width of the opening is mainly increased. The opening may be two to three times larger in area than the geometrically scaled burner. The increased width extends the primary core deeper into the furnace and does not represent the correct location of the combustion front. The velocity profile at a given cross section of a jet depends on the centerline velocity, and the centerline velocity depends on the length of the primary core. The use of enlarged Thring-Newby nozzle results in erroneous velocity profiles as shown in **Figure 3.6** [4]. Zelkowski [5] estimated this error, based on an area method to be 25%, and developed a new model. **Figure 3.7** shows the Zelkowski model.

In the Zelkowski model, the opening enlargement is in between the geometric and Thring-Newby opening. The enlarged opening is recessed from the geometrically scaled location. The area enlargement and the nozzle setback are calculated in terms of the ratio of prototype densities at the nozzle and flame front, the prototype nozzle width, and a characteristic value 'A'. The characteristic value 'A' is the distance between the middle of the windbox (WB) nozzle arrangement to the middle of the convective pass.

Since the inlet burner area for the Zelkowski model is smaller than the Thring-Newby model and operates at the same discharge velocity, the total volume flow through the four burners will decrease. The resulting effect will be an increase in the ratio of the nozzle exit velocity to the mean furnace chamber velocity. Introducing the required volume of additional air through the bottom of the model would lower the velocity ratio to that of the prototype. If the flow pattern area of interest is the whole furnace, the bottom air might disturb the flow patterns in the lower furnace.

The error in lower furnace flow patterns caused by the bottom air might be significantly more than the error by not maintaining the velocity ratio between the model and prototype. If the flow pattern area of interest is only the upper furnace, then a reasonable quantity of air from the bottom can be admitted.

On single and opposed wall firing multiple burner furnaces, using either of the above port enlargement methods can lead to significant error in the momentum flux. As the ports are enlarged the space between openings is reduced. The jets become more confined and entrainment is reduced. This means less flow from Region II enters Region I causing error in the end conditions. The method used to overcome this difficulty is called the "Gauze" method [6]. This technique uses a geometrically scaled burner opening. At the position of the combustion front, a flow resistance is introduced in the form of a gauze or wire screen to produce mechanical drag. The momentum flux is reduced to the correct level while not interfering with the initial jet entrainment and primary core. The end conditions for Region II are thus better simulated. **Figure 3.8** indicates a comparison of burner opening for geometric, Thring-Newby, Zelkowski and gauze methods of furnace burner scaling.

In advanced furnaces utilizing external staging systems to reduce oxides of nitrogen formation, the ports are geometrically scaled. No distortion is done because there is no rapid change in density. The ports are normally located in Region II above the top row of burners. Although the jet is at a lower temperature than furnace gases, its density drops gradually as it penetrates and mixes. In the model, the jet is entraining air of equal temperature and can be expected to give a conservative estimate of jet penetration. The prototype external staging jets penetrate to a greater distance than that observed in the model.

Furnace Model Flow Evaluation Methodology

The furnace model flow evaluation approach uses both qualitative and quantitative methods. The qualitative method reveals the gross flow patterns for basic understanding, and locates the problematic areas quickly. The quantitative method reveals accurate detailed flow patterns, and quantifies the velocities.

The qualitative evaluation uses flow visualization with yarn streamers, smoke and helium bubbles. Inserting a yarn tuft tied to the tip of a rod into the flow field can show the flow direction. Admitting white smoke through the windbox nozzles reveals the gross flow patterns. Titanium tetrachloride smoke sticks generate the smoke.

Injecting neutrally buoyant helium filed soap bubbles into.the air flow stream visualizes the flow in detail. The bubbles follow the flow streamlines. A Sage Action Model 33 Multi-head Bubble Generator assembly generates the bubbles. Videotapes and photographs of the smoke tracer flow patterns, and the helium bubble streak lines at selected flow conditions with proper illumination of the model are used for future evaluation.

The quantitative evaluation uses a five hole pitot tube for determining the velocity vectors. The pitot tube has five faces each having a hole. When face 1 is aligned with the flow, the static pressures on faces 2 and 3 are equal. The differential pressures between the holes on faces 1 and 2, 1 and 4, and 4 and 5 are used to calculate the total velocity vector, and the x, y, and z velocity components. The velocity vectors are color plotted.

The flow patterns, and velocity magnitudes at various locations of the furnace are evaluated. The basis of comparisons are:

(1) The general model flow patterns.

(2) The velocity levels between furnace configurations.

(3) The average velocity ratio between the resultant of XY velocity components to the Z-direction velocity.

The first basis ensures whether the general model arrangement is meaningful according to a logically expected model. The second basis would reveal the relative furnace residence times between furnace configurations for evaluating the carbon burnout. The third basis tells the residence time at various elevations of each furnace arrangement for evaluating the effect on NOx emission, and furnace slagging and corrosion problems. The higher the velocity ratio, greater the degree of rotation and residence time.

RILEY STOKER'S CORNER-FIRED PHYSICAL FLOW MODEL STUDY

To expand Riley Stoker's in-house knowledge, support retrofit efforts, and document the aerodynamic flow patterns, a 1/12th scale model testing program began. A furnace with an aspect ratio of 1.43 was selected for the three dimensional, isothermal corner-fired furnace physical modeling.

The model scope included three configurations of corner windbox assemblies; the furnace; the furnace arches; and the backpass entry. No convective heat exchanger surfaces were modeled. The three windbox configurations were:

(1) Original corner-firing windbox(Pre-NSPS firing system)

(2) All corners auxiliary air directed along the wall

(3) Opposite corners auxiliary air directed along the wall

Figure 3.9 shows the plan and 3-D views of the three configurations respectively. The original corner windboxes were 16" wide and separated into two sections.

The second windbox arrangement used the gap between the separate upper and lower portions of the original windbox. The new corner windbox was horizontally 2" wider than the original. All four corner windboxes had the auxiliary secondary air nozzles horizontally offset by 22° from the coal nozzle firing angle. This arrangement had external staging ports above each corner windbox.

In corner-firing, corners with smaller firing angle with respect to the front or rear wall are called hot corners because the centerline of the flame runs closer to the wall. This reduces gas entrainment due to limited space between the jet and the wall and creates additional drag on this side of the jet.

In the third windbox arrangement, only the hot corners had the offset auxiliary secondary air nozzles. In the cold corners, auxiliary secondary air nozzles and coal nozzles had the same angle with respect to the front wall. This arrangement also used the gap between the separated upper and lower windboxes, and was also horizontally 2" wider. This arrangement used external staging ports above each corner windbox.

The model test objectives were:

Qualitative visual evaluation of the flow patterns in the upper furnace and along the wall using yarn streamers, smoke, and helium bubbles;

Quantitative three dimensional velocity mapping data determination using a five hole Pitot tube;

Validation of the FLUENT numerical computer model output using the physical model's velocity mapping data.

The model engineering, design, construction and testing took place at the Riley Research Center in Worcester, Massachusetts. The furnace was geometrically scaled. The coal and air nozzles were scaled using the Zelkowski burner modeling technique. The size of the existing outlet ductwork limited the amount of air flow that could be run through the model. However, a Reynolds number higher than the minimum value required for the model and prototype flow field similarity (10,000) was maintained, and the flow was adequate to document and visualize the general flow conditions in the furnace.

Plexiglass (3/8" thick), wood, sheet metal and angle iron were used for model construction. The model was tested under suction similar to balanced draft conditions. **Figure 3.10** shows a photo of the model. The testing included flow visualization using smoke sticks and helium bubbles, videotaping, and velocity traversing using a five hole and standard Pitot tubes. Data reduction included calculation of the total, X, Y, and Z direction velocities at each traverse point from the pitot traverse data. Data analysis was done by comparing and evaluating the flow patterns and velocity magnitudes between the three furnace configurations and at various planes of each furnace.

<u>The test results and conclusions were:</u>

1) For all three windbox configurations, the opposite corner nozzle velocities were at similar level. Cold corner (front-right and rear-left) velocities were slightly higher than the hot corner velocities. This is attributed to drag on jets closer to the walls and the 1.43 aspect ratio of the furnace plan.

2) In the third arrangement, corners with offset auxiliary secondary air had less flow than the corners without the offset. The lower flow was due to the higher resistance caused by the offset than the nozzle without the offset. In the field, the corner maldistribution could be corrected by partially closing the cold corner dampers.

3) In general the visualized flow patterns and measured velocity vectors verified the expected flow patterns for each of the

corner-firing configurations. Figure **3.11** shows the freehand sketch of flow patterns in the furnace for the three windbox configurations. **Figures 3.12 through 3.14** show the traversed velocity vector plots on XY surface at Planes 1, 4 and 6, respectively and traversed Z direction velocities at corresponding Planes of the three corner-firing configurations. The following paragraphs describe in general the velocity vector plots; and the Z direction velocity surface plots.

Various colors in the velocity vector plots show various grid points' true total velocity magnitude. The vectors are 3-D entities(the resultant of X, Y, Z direction velocity components) projected onto a 2-D surface(XY Plane). The head on each velocity vector shows the flow direction. If a vector is perpendicular to the paper, then only a cross for the arrow head shows, but the color still gives the velocity magnitude. It does not show whether the flow is up or down. The Z-direction velocity surface plots indicate the upward and/or downward flow regions in the furnace at their respective planes.

All velocity vector plots use the same linear color scale to aid in comparisons between configurations. Eight colors in the plots correspond to eight velocity magnitude ranges with the central values shown in the key. The key values of black 90 ft/sec, violet 78 ft/sec, and green 30 ft/sec mean that a violet vector has a velocity magnitude in the range of 72 ft/sec to 84 ft/sec no matter how long or short it looks. Finer resolution on velocity magnitude would require more color.

The following observations were made from the visual and velocity vector plots (**see Figures 3.11 through 3.14**):

Flow oscillation was observed in the upper furnace for all three configurations. This was significant in the original corner firing compared to the other two configurations;

A counter clockwise rotating vortex is obvious;

Velocities near the wall regions are high and decrease towards the furnace center. This is alright because it is due to the effect of the rotating vortex's centrifugal force, and the jets;

The maximum velocities are at the middle region of front and rear walls. This might be due to the compression of the free circular vortex on the hot corner jets, and higher aspect ratio(1.43);

The total velocity magnitude decreases from bottom plane to top plane despite the increasing mass from bottom to top. This might be due to: (i) larger free flow area due to the flow spread in the upper furnace, and (ii) Drag between wall and flow streams and between flow streams.

Except at plane 1(middle of the bottom end compartment), the furnace flow field is moving upward.

At Plane 1, the flow field near the walls directs downward, and comes up in the center region.

4) At similar inlet flow rates, the second and third configurations had lower model inlet and furnace velocities than original corner-firing. The third configuration furnace velocities were slightly lower than the second configuration furnace velocities. The lower velocities would increase the overall residence time in the furnace and provide an opportunity for good carbon burnout.

At each grid point of a plane, the ratio of the resultant of the X and Y direction velocities to the Z-direction velocities (Vr(xy)/Vz) was calculated. The velocity ratios were averaged for each plane. This average ratio indicates the degree of rotation and the residence time in the plane considered. In a given plane, the higher this ratio, the greater the horizontal swirl, and the longer the residence times. The other factor that should be considered is the actual velocity distribution. Lower Z-direction velocity increases the residence time in the furnace. Higher XY plane resultant velocity increases the degree of rotation.

5) **Figure 3.15** compares the average velocity ratio along the height of the furnace for the three configurations. The lower furnace had a higher degree of rotation and residence time than the upper furnace for the three cases studied.

In the upper furnace, the velocity ratio and thus the degree of rotation and residence time were higher for original corner-firing than the second and third configurations. Inthe lower furnace, the degree of rotation was at similar level for the three configurations. In a substoichiometric atmosphere(the second and third configurations), these conditions would act to improve stability, and devolatilization and lead to lower overall NOx emissions through potential NOx destruction reactions.

6) Due to increased O2 availability near the walls, slagging and corrosion potentials would be reduced for the second and third configurations.

7) The FLUENT computer model's overall velocity vector mapping was similar to physical model's vector mapping. However, due to model scaling choices, the FLUENT inlet size did not correspond to the physical model, and thus the mass flow and absolute velocities were different.

FUTURE WORK

Riley's future work includes model testing with various loads, different air distributions, different locations and trajectories of external staging, changes to aspect ratio and improved data acquisition with longer five hole Pitot tube, more detailed velocity mapping, and a detailed FLUENT study with similar inlet conditions between the FLUENT and physical models.

REFERENCE:

[1] C.E. McHale, R.A. Lisauskas, "The Application Of The Zone And Monte Carlo Methods To Predict Heat Transfer In Pulverized Coal Fired Utility Boilers", ASME Winter Annual Meeting, Boston, MA., November 13-18, 1983.

[2] Davison, F.J., "Nozzle Scaling in Isothermal Furnace Models", Journal of the Institute of Fuel, December 1968.

[3] Beer, J. M. and Chigier, N. A., "Combustion Aerodynamics", Wiley and Sons, Inc., New York, 1972.

[4] Thring - Newby Papers

Pierce, T.J. and Thring, M.W., "Distribution of Residence Times in a Model of a Pulverized Fuel Boiler", Proceeding Second Conference on Pulverized Fuel, Institute of Fuel, London, 1957.

Johnstone, R.E. and Thring, M.W., "Pilot Plant Models and Scale-Up Methods in Chemical Engineering", McGraw Hill Book Company, New York, 1957

Thring, M.W. and Newby, M.P., "Combustion Length of Enclosed Turbulent Jet Flames", Fourth International Symposium on Combustion, Williams and Wilkins, Baltimore, MD, 1953, p. 789.

[5] Zelkowski, V. J. Warschau (polen), "Die isothermische Modellierung der Stromung in Brennkammern von Staubkesseln mit Eckenbrennern", ("The Isothermal Model of Flow in Combustion Chambers of Pulverized Coal-Fired Boilers with Corner Burners"), Mitteilugen der VGB, No. 104: 335-344 (October 1966)

[6] Anson, D., "Modeling Experience with large Boilers", Journal of the Institute of Fuel, January, 1967 (Gauze Technique)

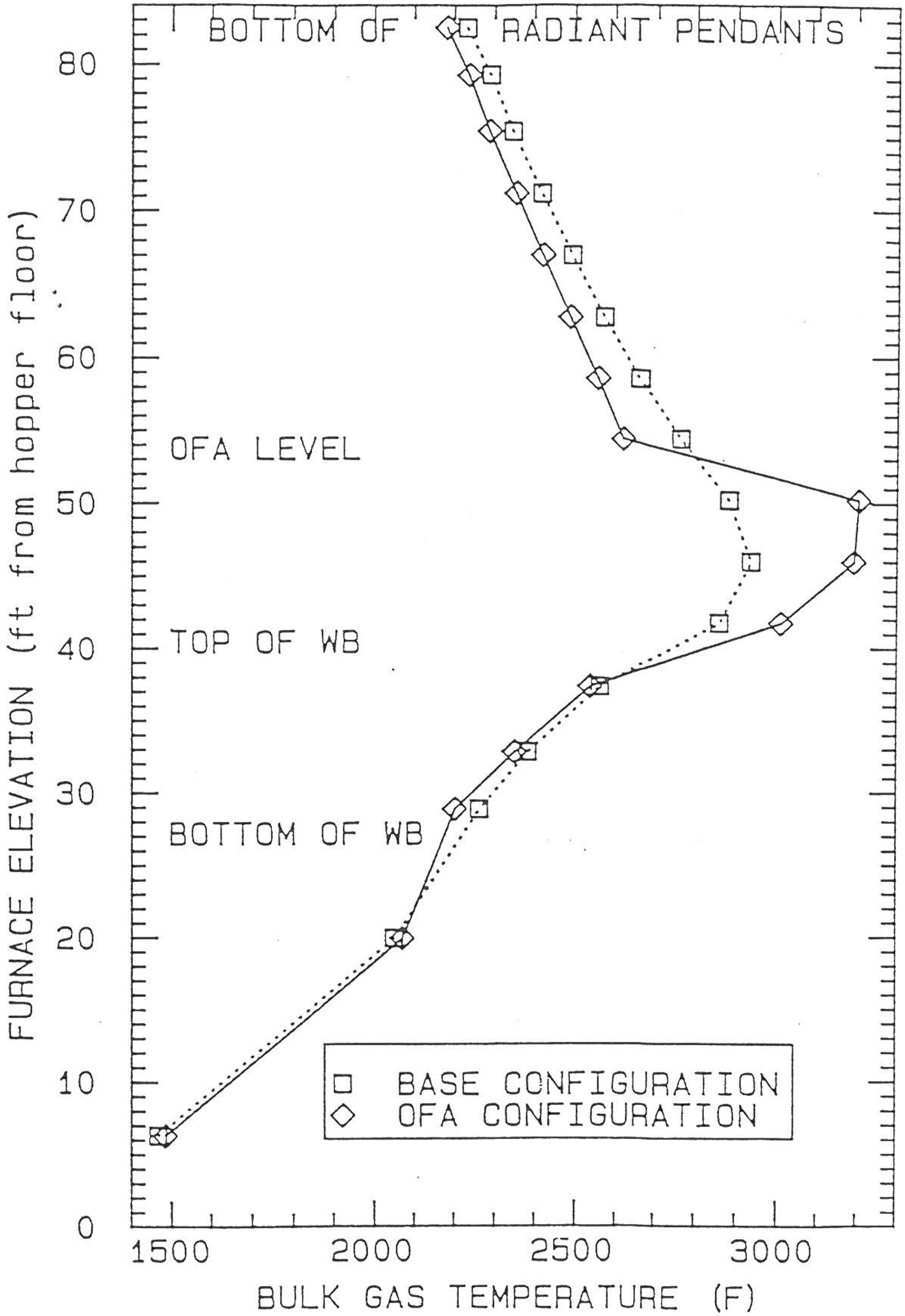

Figure 1.1 FASTFIRE OUTPUT FOR PULVERIZED COAL CORNER FIRED FURNACE

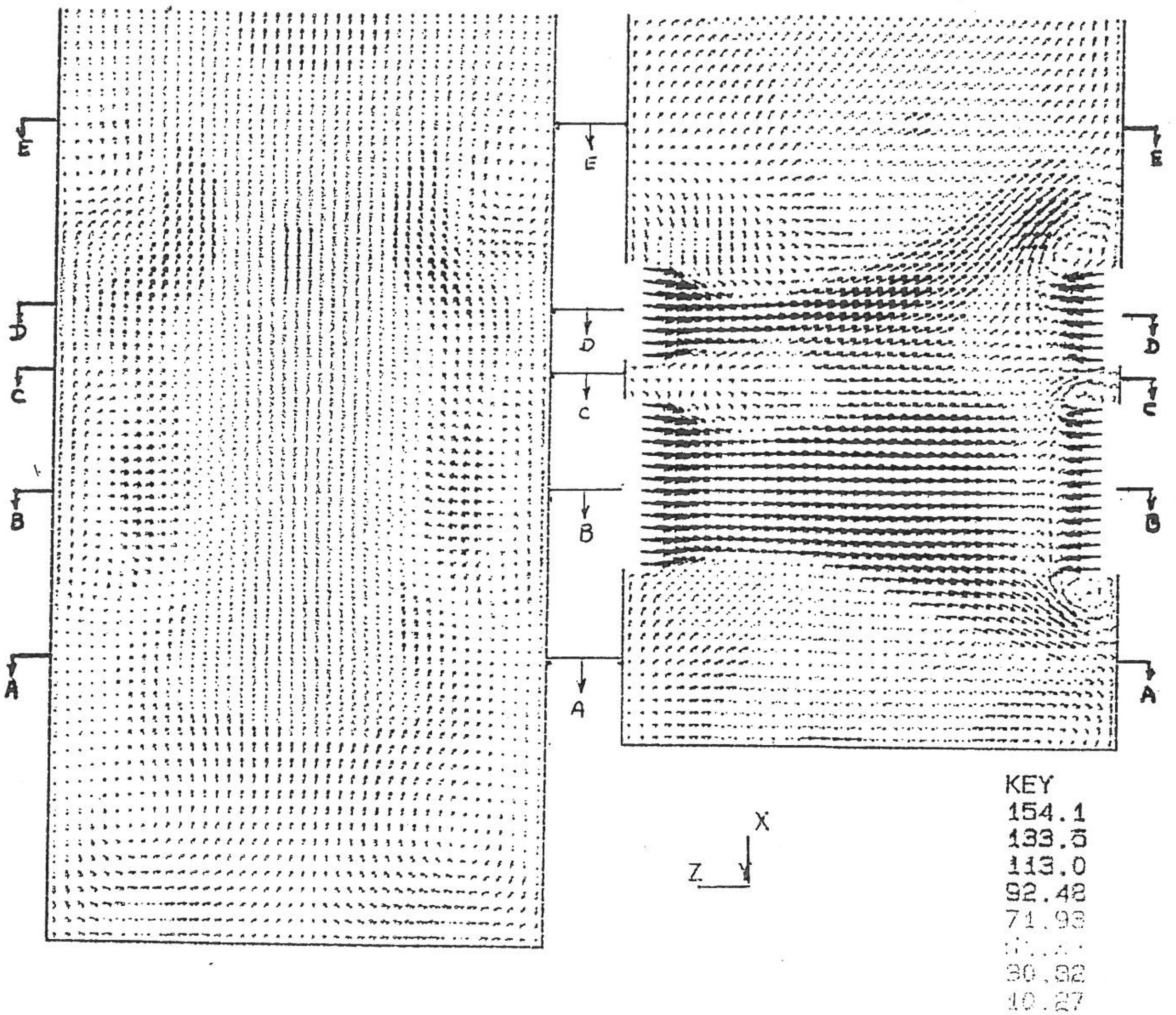

Figure 2.1 BASE CASE VELOCITY VECTORS (FEET/SEC)
AT TWO CROSS SECTIONS

KEY
154.1
133.5
113.0
92.48
71.93
51.38
30.82
10.27

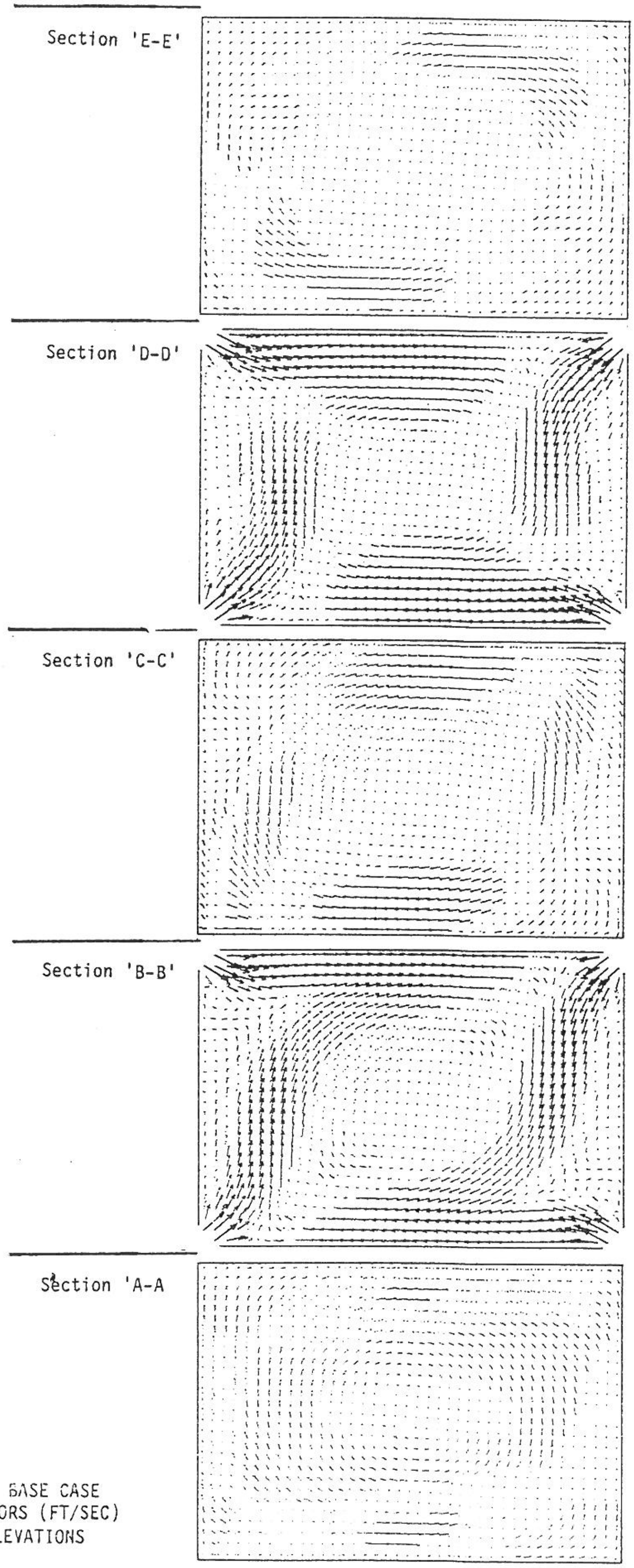

FIGURE 2.2 BASE CASE
VELOCITY VECTORS (FT/SEC)
AT FIVE ELEVATIONS

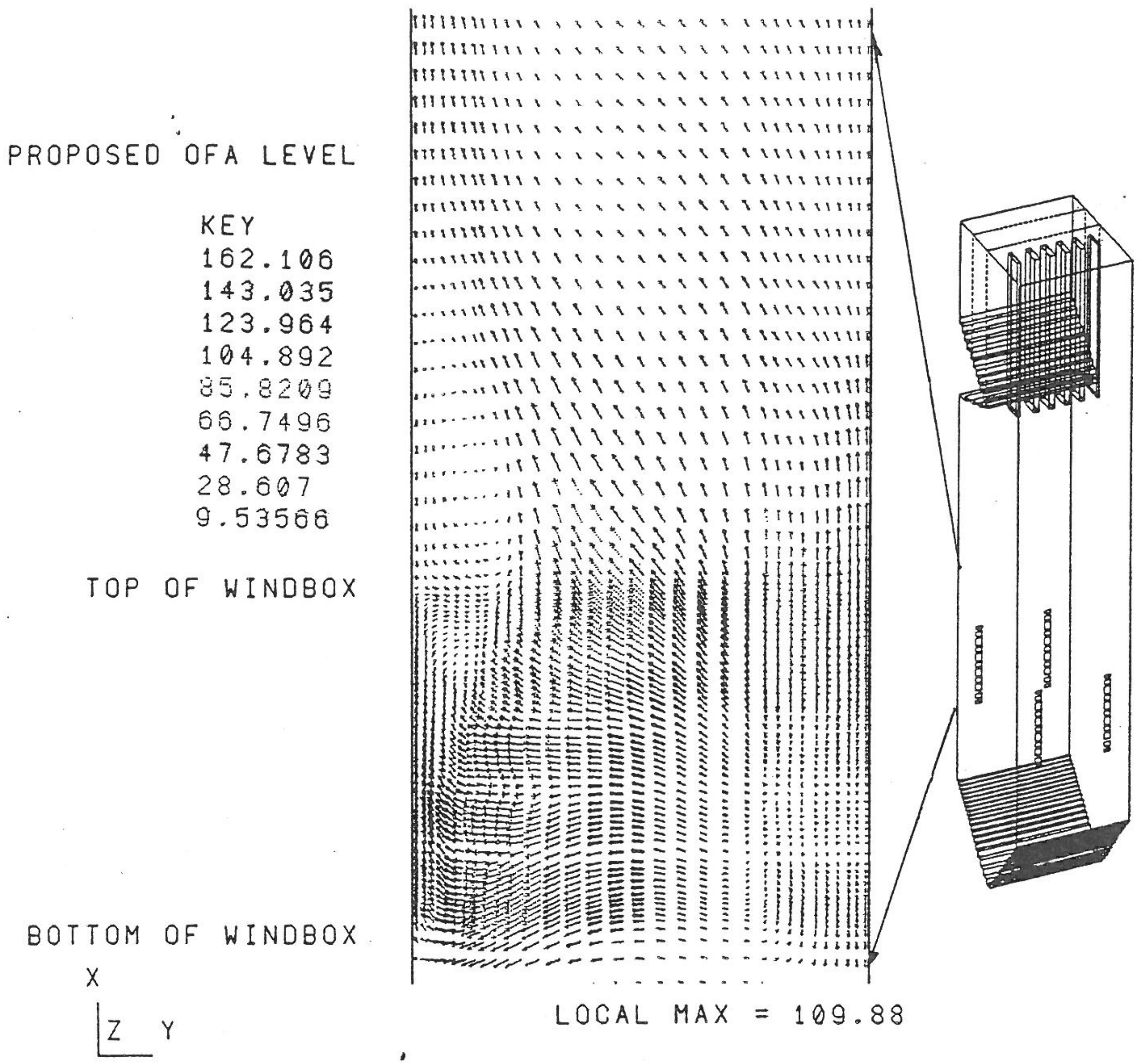

FIGURE 2.3 DIVIDED FURNACE BASE CASE
VELOCITY VECTORS (FEET/SEC) AT 8 FEET FROM REAR WALL

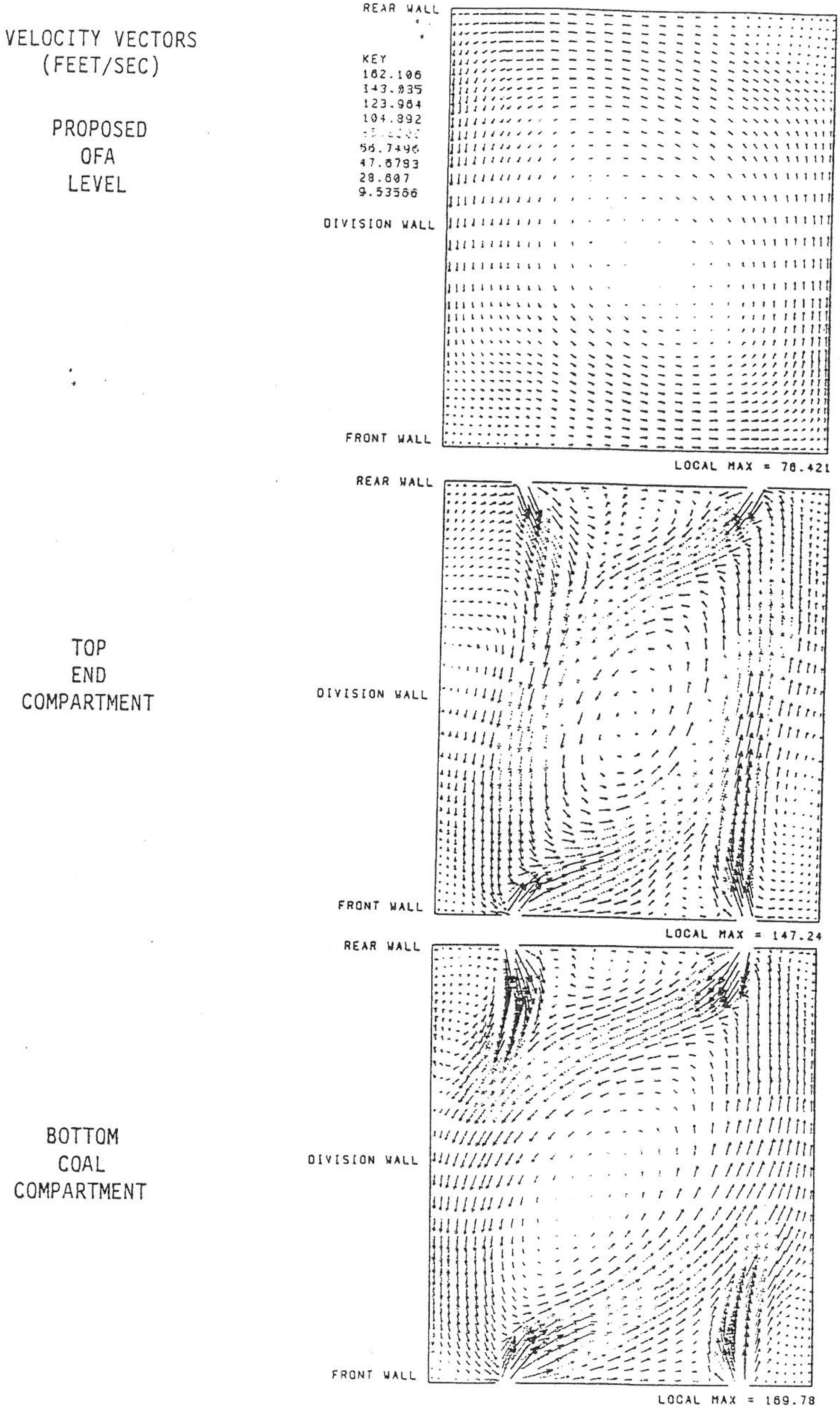

FIGURE 2.4 DIVIDED FURNACE BASE CASE
VELOCITY VECTORS (FEET/SEC) AT THREE ELEVATIONS

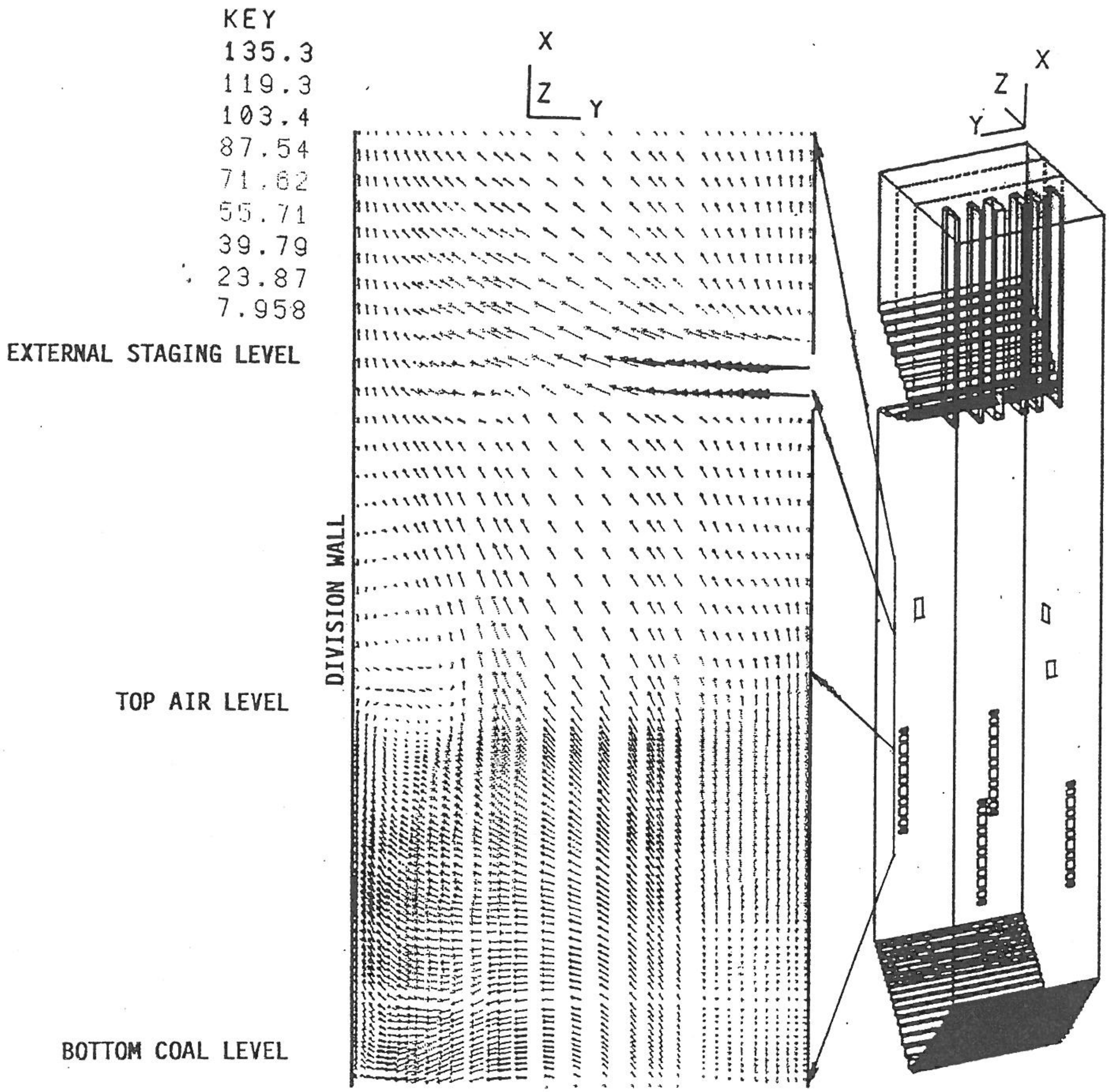

Figure 2.5 DIVIDED FURNACE WITH EXTERNAL STAGING, VELOCITY VECTORS (FEET/SEC) AT 8 FEET FROM REAR WALL

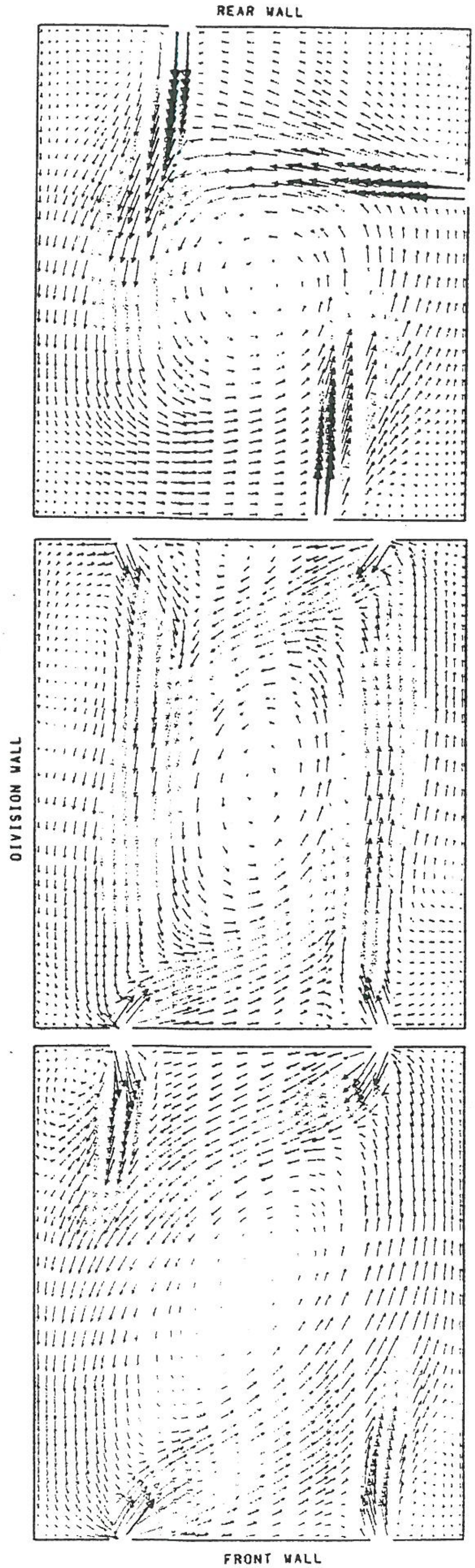

Figure 2.6 DIVIDED FURNACE WITH EXTERNAL STAGING, VELOCITY VECTORS (FEET/SEC) AT THREE ELEVATIONS

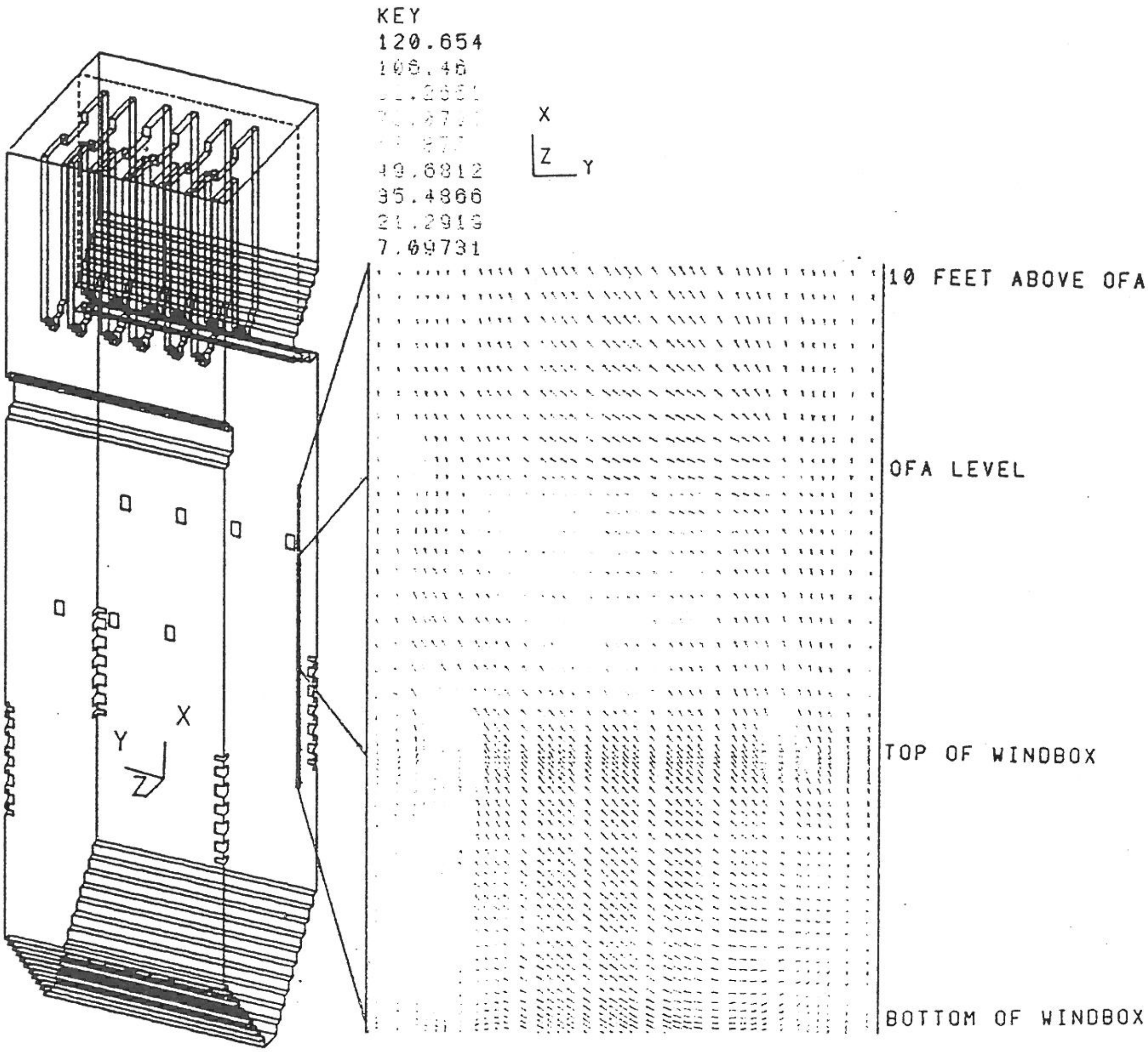

Figure 2.7 GEOMETRIC OUTLINE AND VELOCITY VECTORS (FEET/SEC) 7.5 FEET FROM REAR WALL

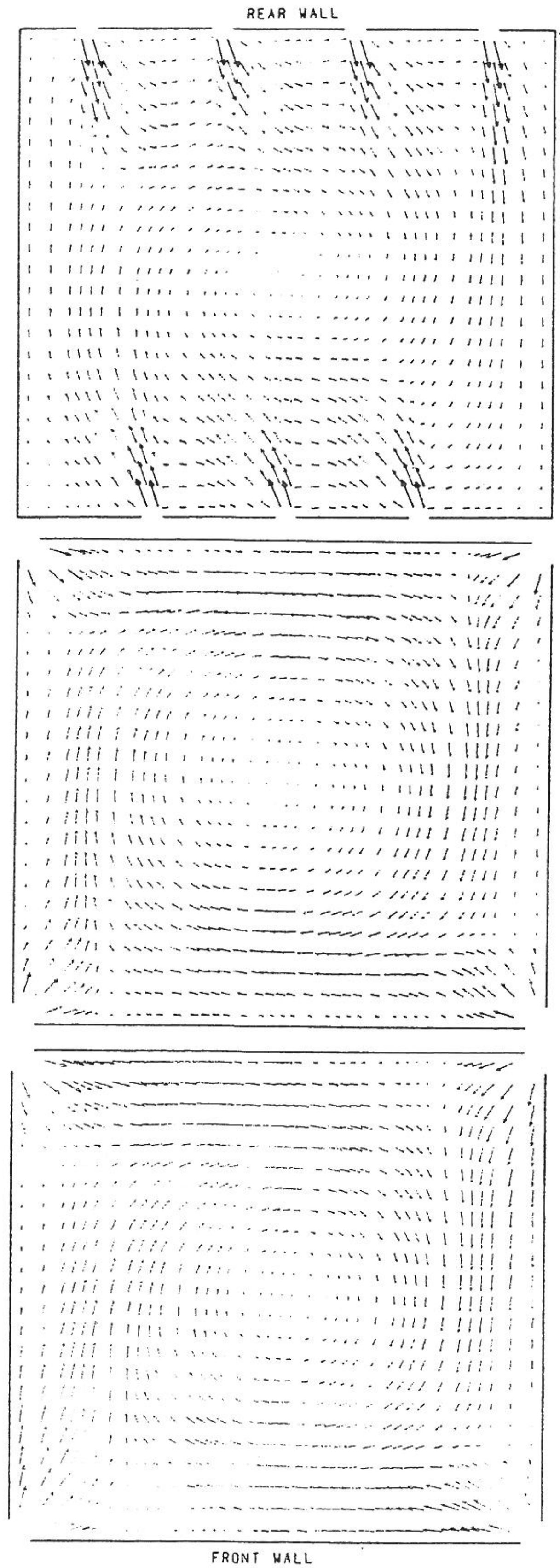

Figure 2.8 FRONT/REAR WALL EXTERNAL STAGING VELOCITY VECTORS (FEET/SEC) AT THREE ELEVATIONS

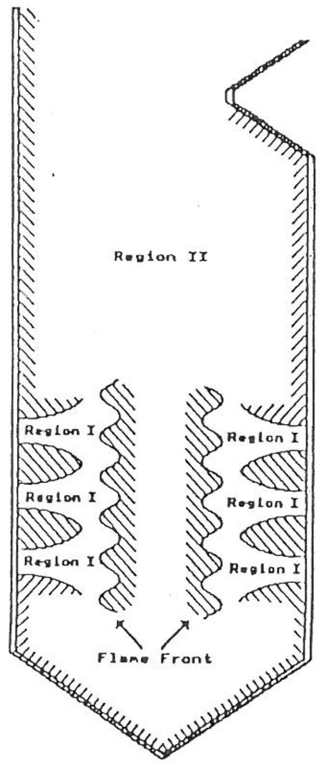

Figure 3.1
Simplified Isothermal
Furnace Flow Model

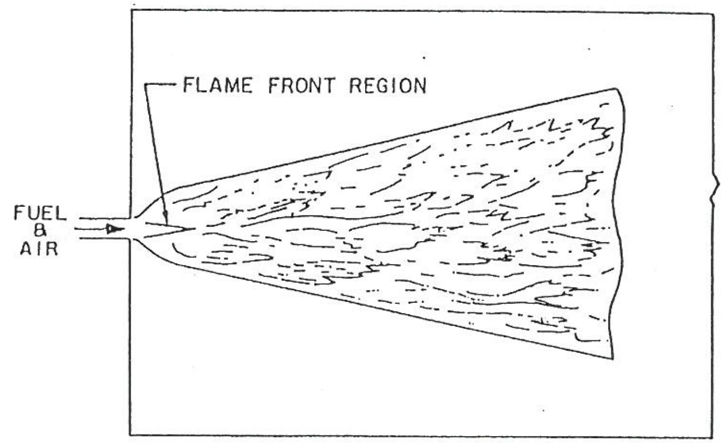

Figure 3.2 Single Burner Free Jet Under Combustion Conditions

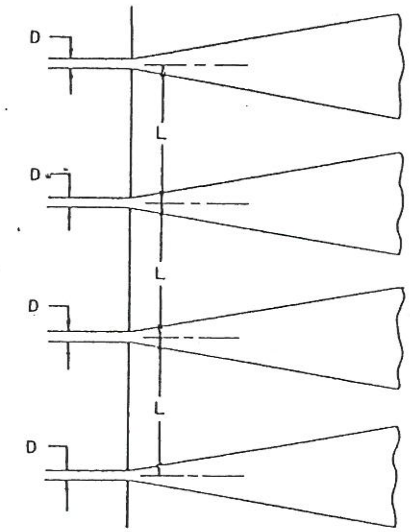

Figure 3.4 Multi Burner Flame Jets With L/D > 18

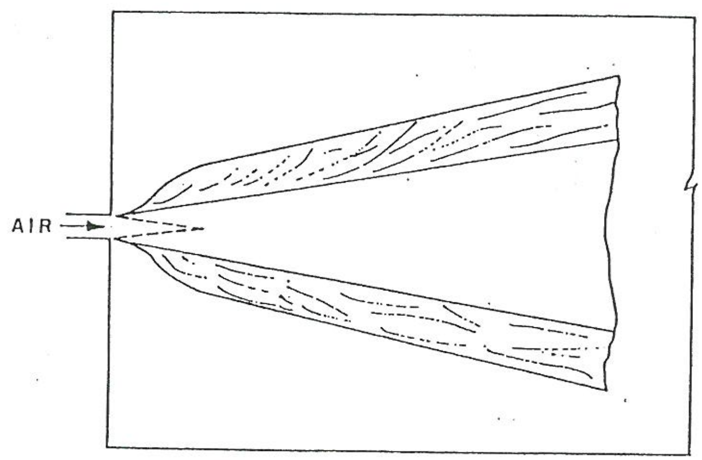

Figure 3.3 Isothermal Free Jet Superimposed On The Combustion Free Jet

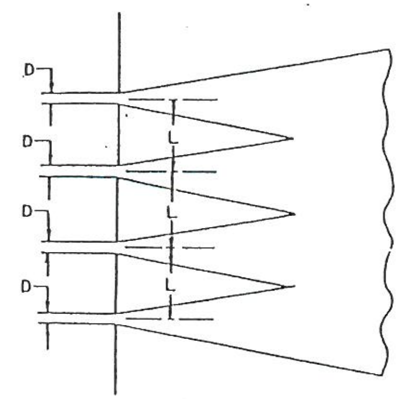

Figure 3.5 Multi Burner Flame Jets With L/D < 18

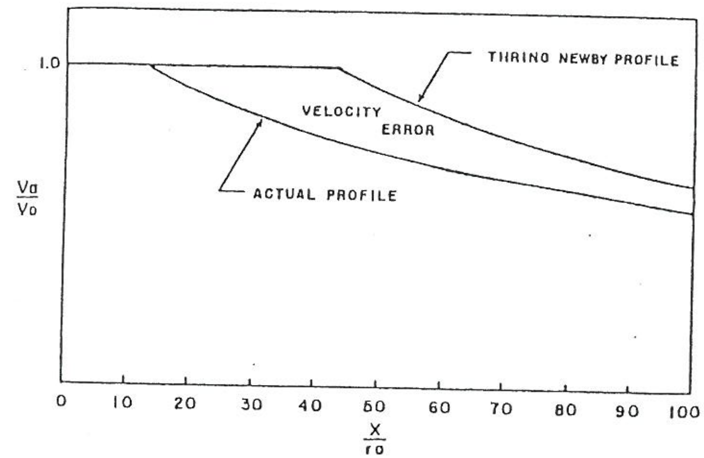

Figure 3.6 Velocity Error In Thring Newby Velocity Profile

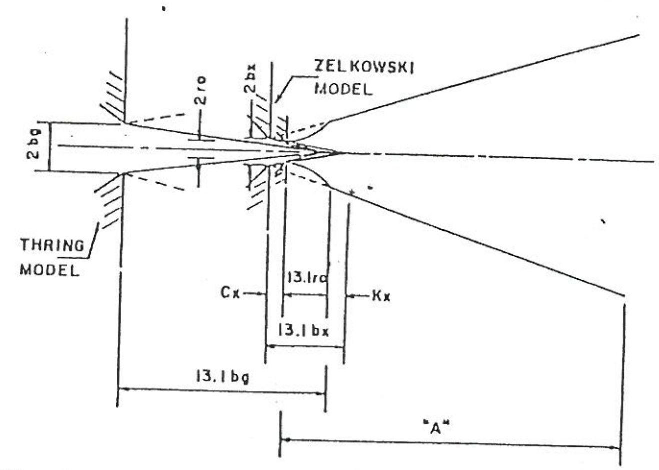

Figure 3.7 Zelkowsky Model Parameters

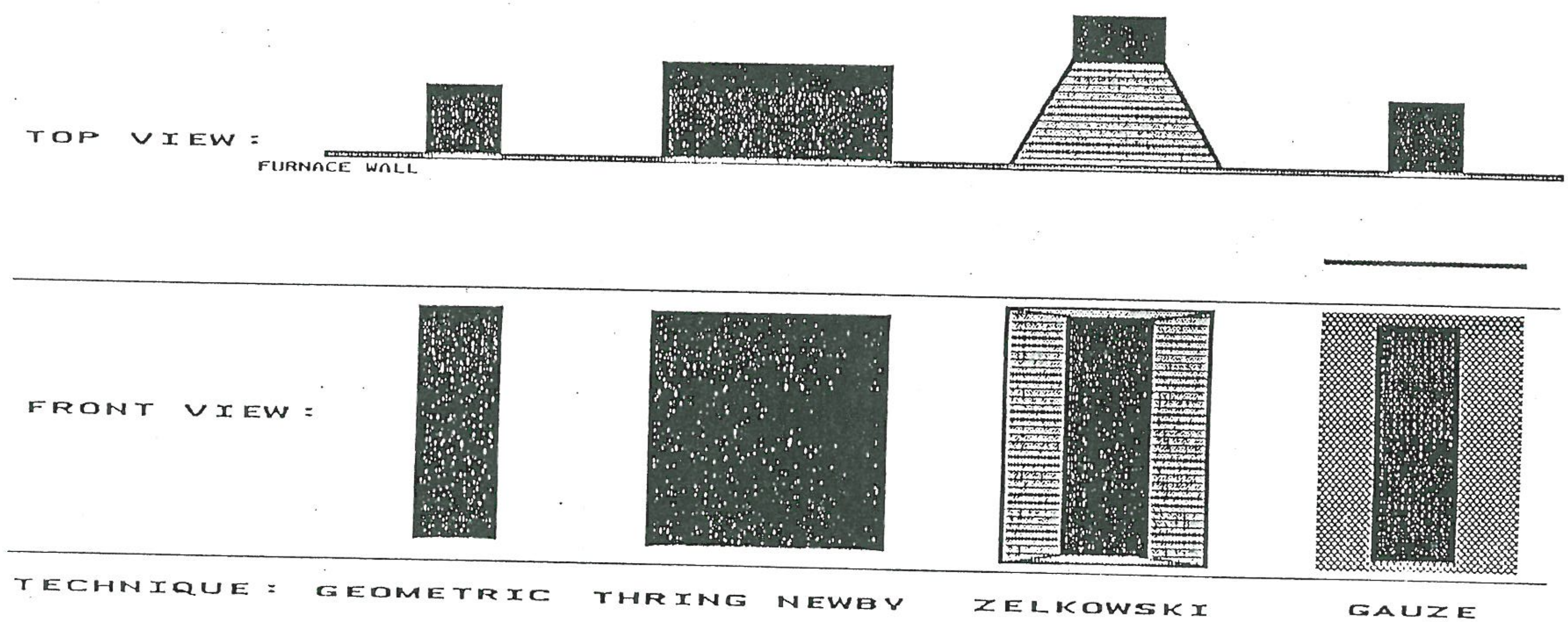

Figure 3.8 Comparison Of Various Modeling Techniques

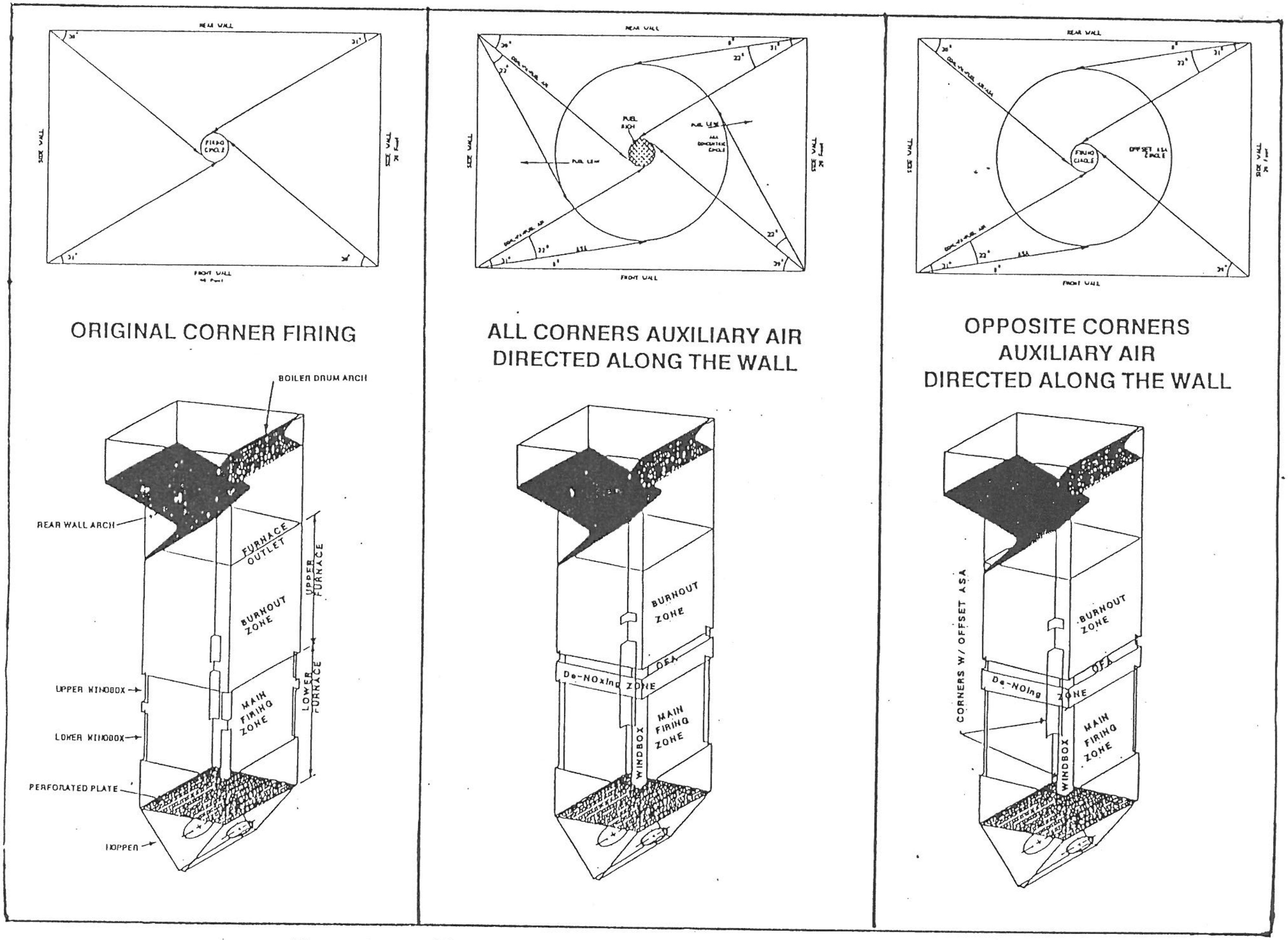

Figure 3.9 Plan And 3-D Views Of The Three Corner-fired Furnaces

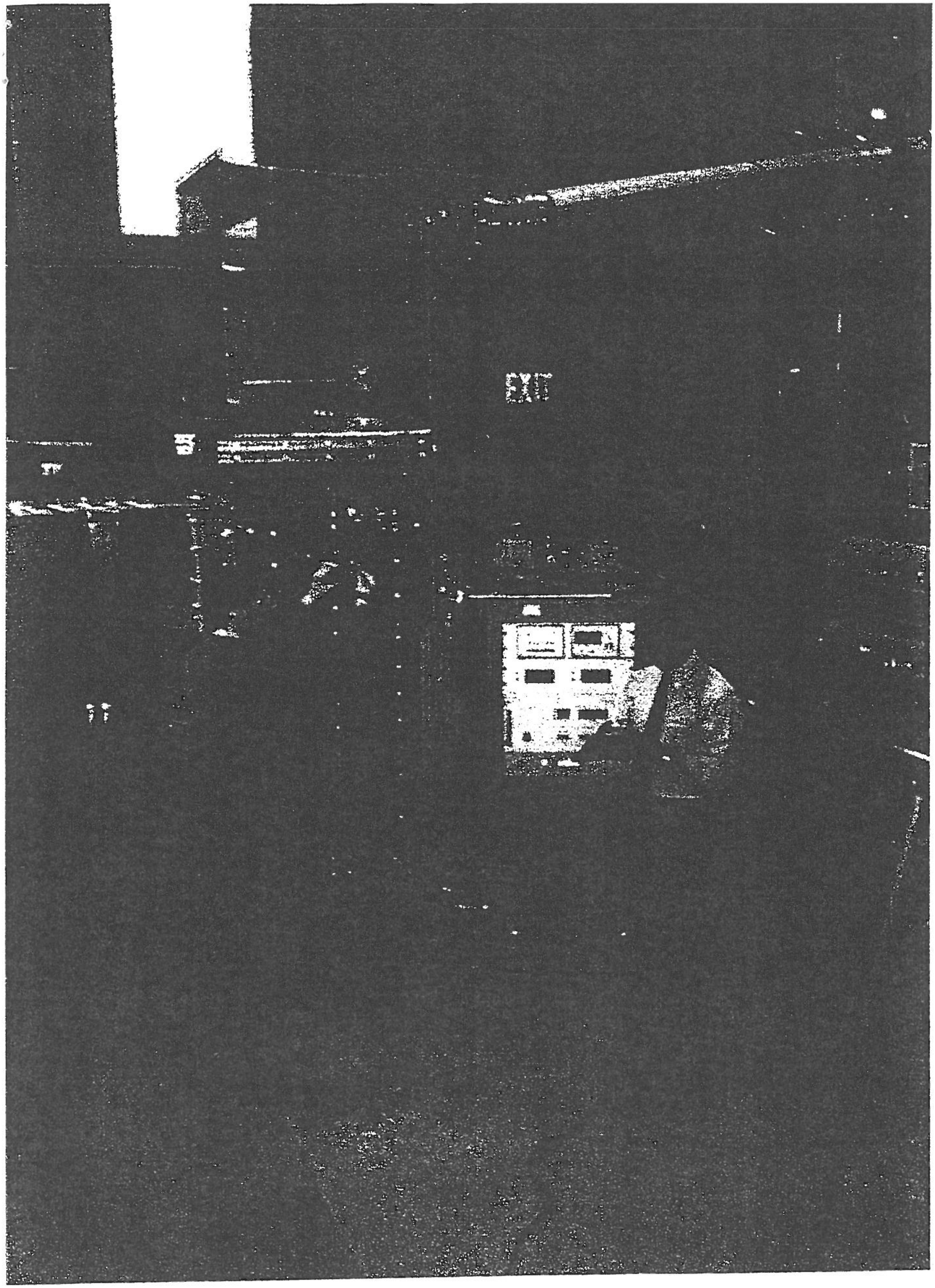

Figure 3.10 Photo Of Riley's 1/12 Scale, Corner-fired Furnace Physical Flow Model

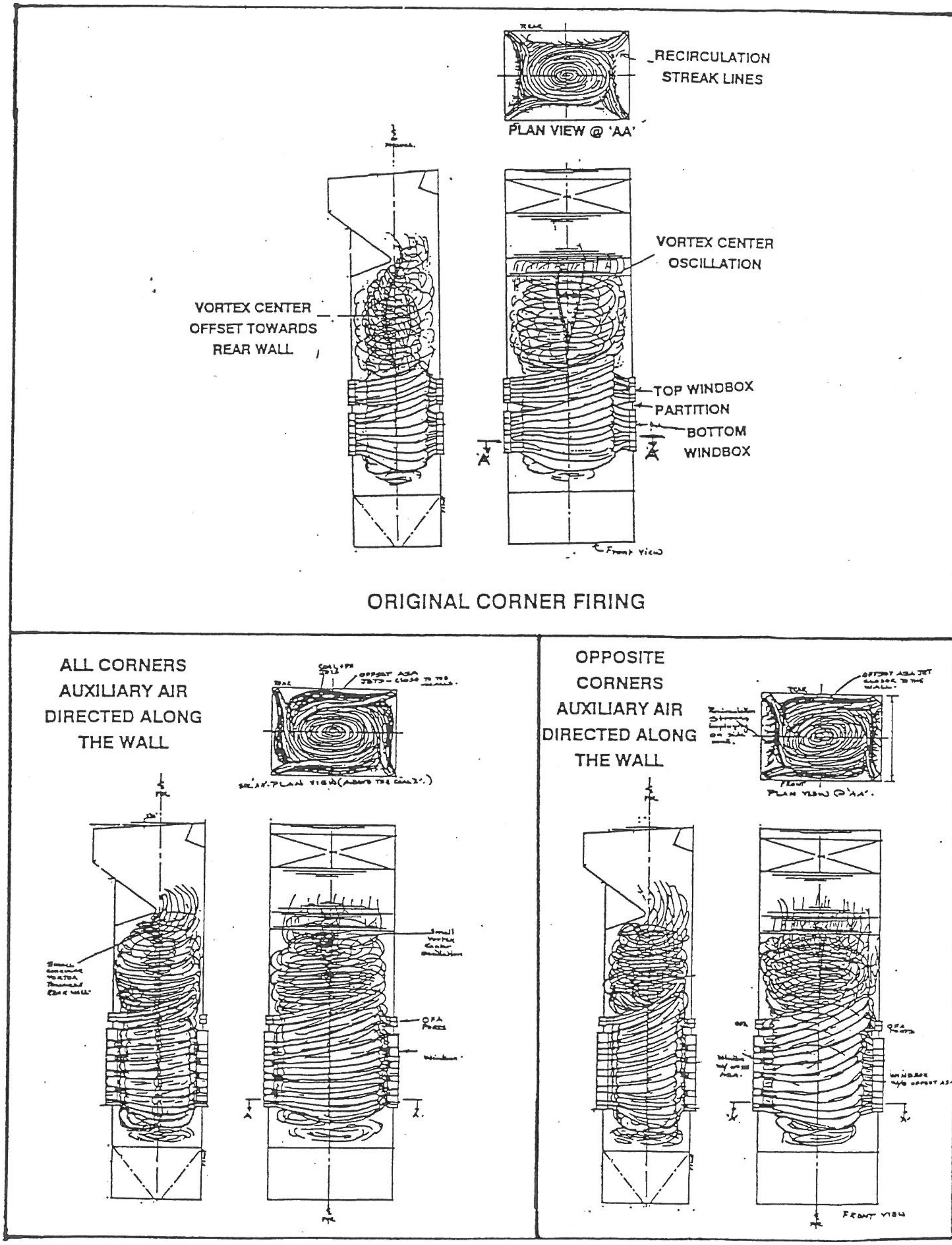

Figure 3.11 Free Hand Sketches Of Flow Patterns In The Furnace

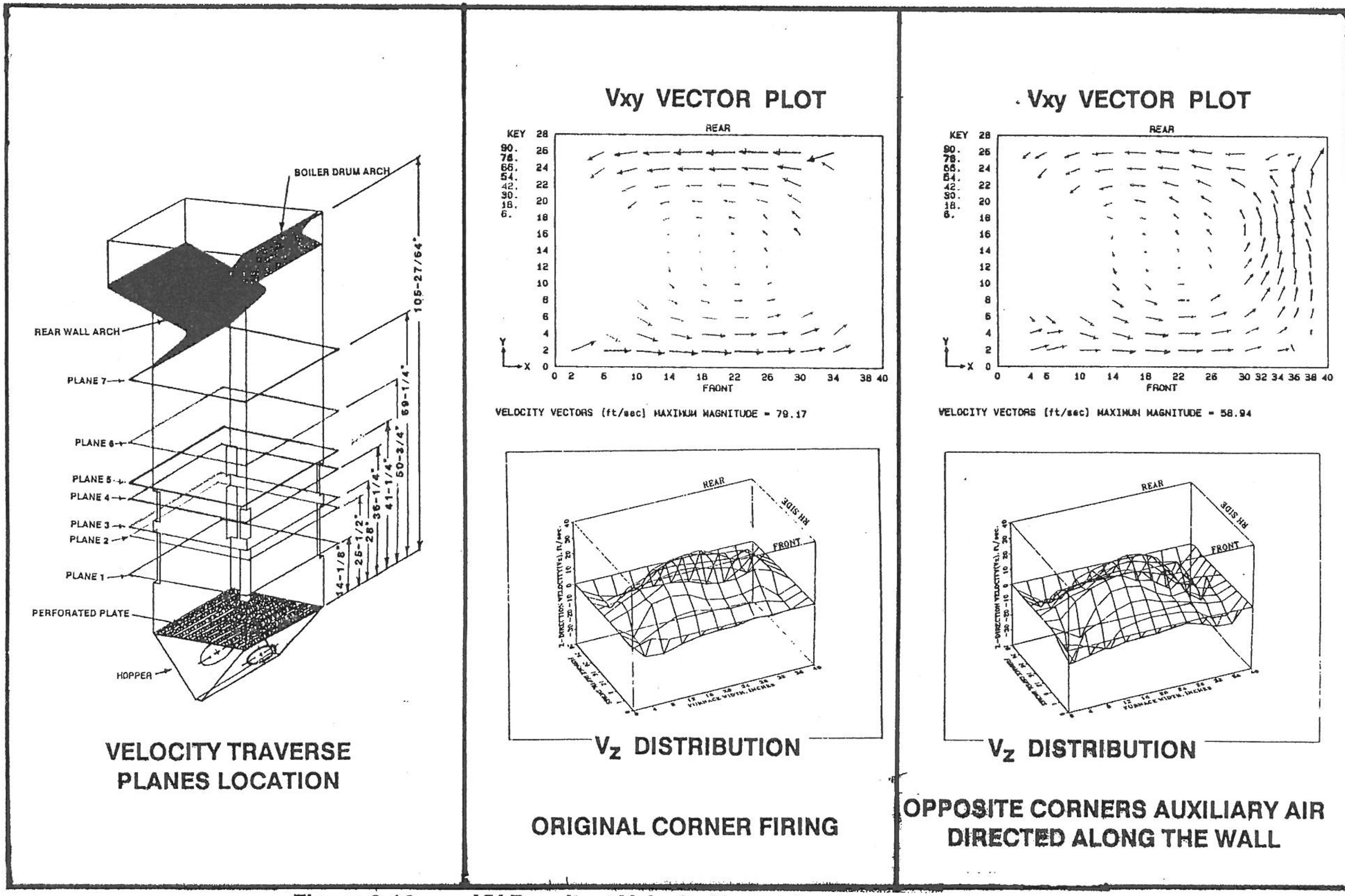

Figure 3.12 XY Resultant Velocity Vector And Z-direction Velocity Distribution In The Lower Furnace(Plane 1)

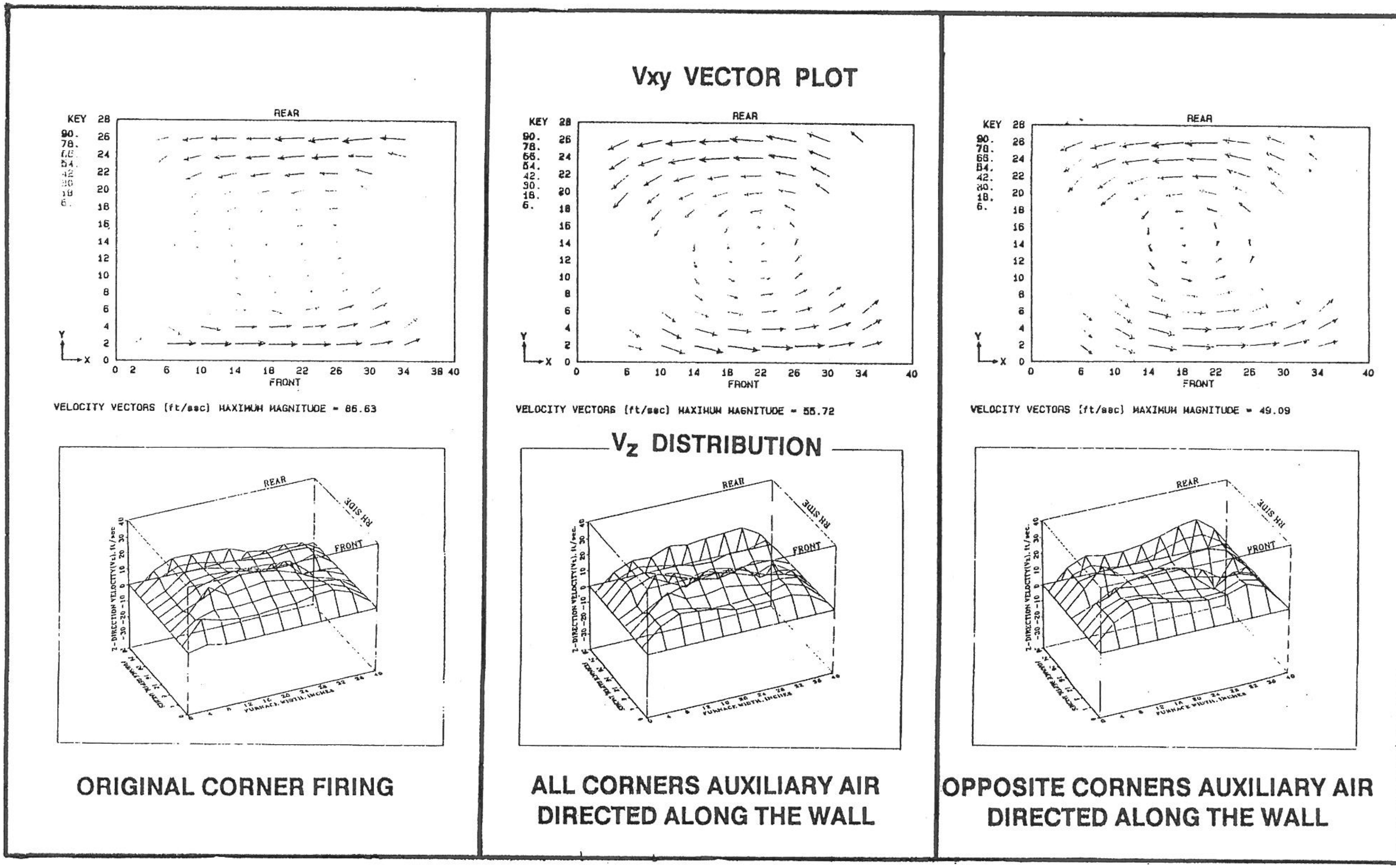

Figure 3.13 XY Resultant Velocity Vector And Z-direction Velocity Distribution In The Mid Furnace(Planes 4 & 5)

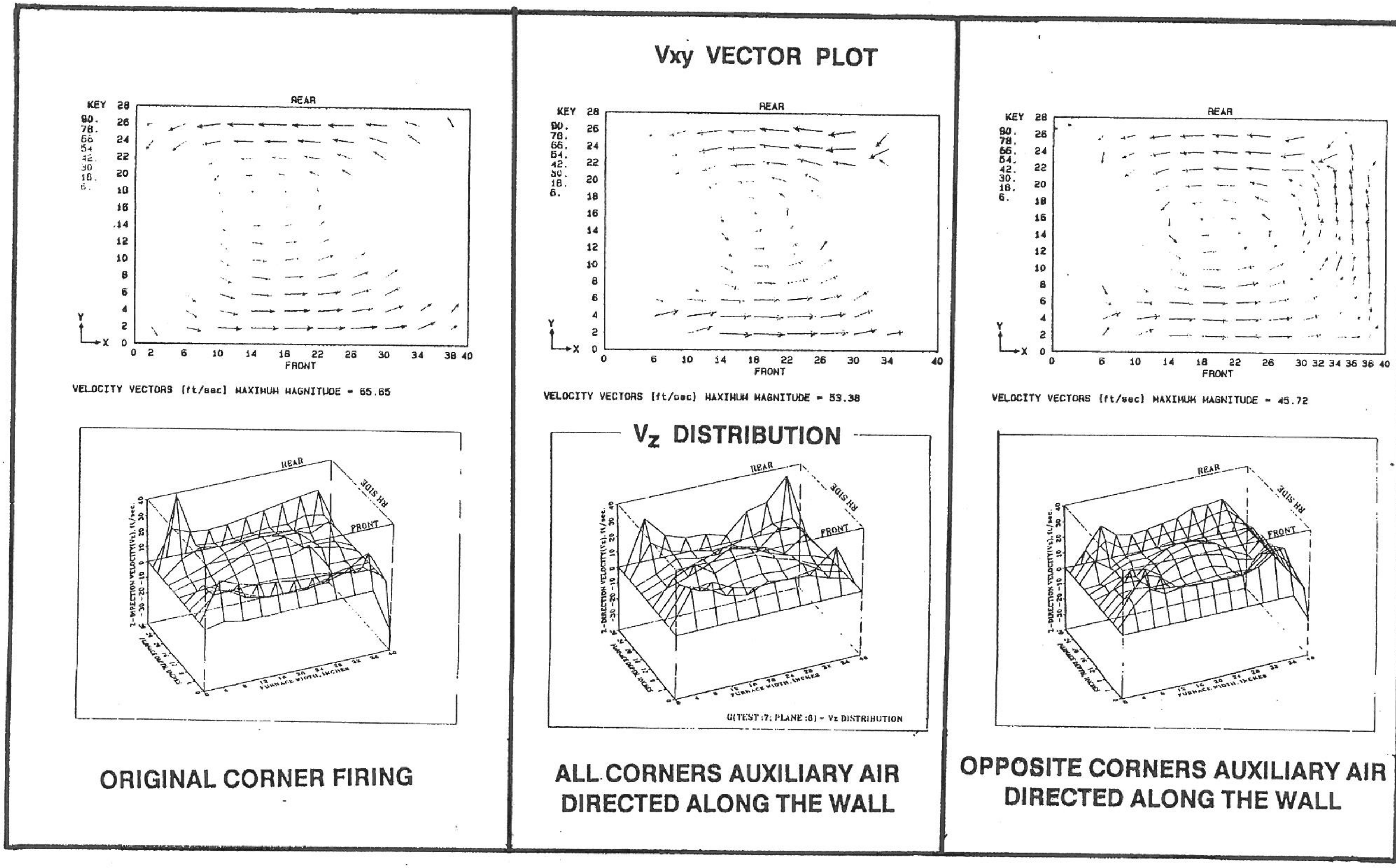

Figure 3.14 XY Resultant Velocity Vector And Z-direction Velocity Distribution In The Upper Furnace(Plane 6)

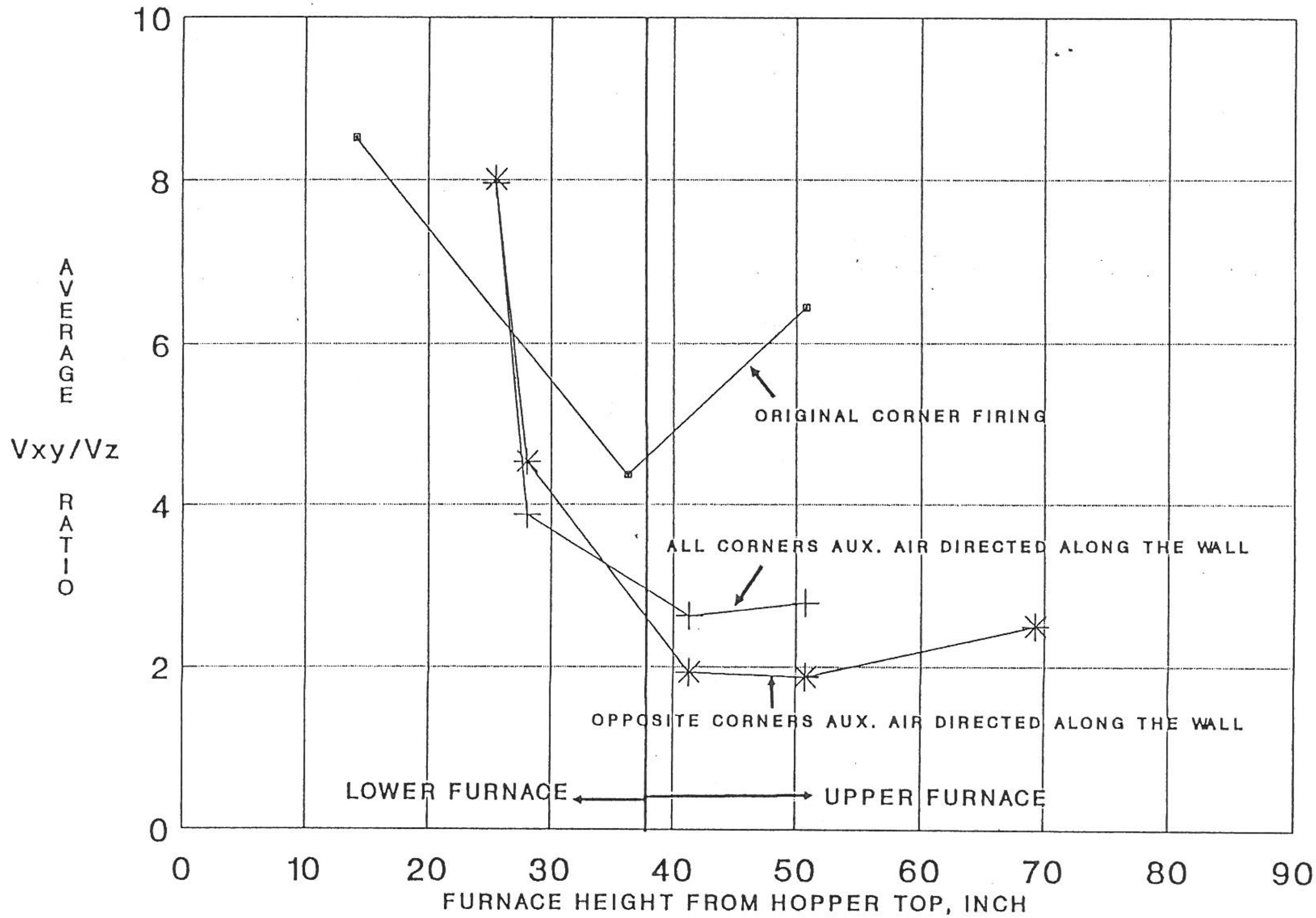

Figure 3.15 Comparison of Average Velocity Ratio