
10 Burner Physical Modeling

John P. Guarco and Tom Eldredge, Ph.D

CONTENTS

- 10.1 Introduction
- 10.2 Methods of Dimensional Analysis
 - 10.2.1 Buckingham Pi Theorem
 - 10.2.2 Method for Determining the Π Groups
 - 10.2.3 Application of the Buckingham Pi Theorem
- 10.3 Dimensional Analysis and Model Studies
- 10.4 Airflow Requirements for Proper Burner Operation
- 10.5 Using Scaled Modeling for Achieving Proper Burner Airflow
- 10.6 Characterization of Burner Swirl
- 10.7 Techniques for Burner Modeling
 - 10.7.1 The Thring-Newby Method for Burner Modeling
 - 10.7.2 The Zelkowski Method for Burner Modeling
 - 10.7.3 The Davidson (Gauze) Method for Burner Modeling
 - 10.7.3.1 Case A: Free Jet
 - 10.7.3.2 Case B: Confined Jet
- 10.8 Scaled Modeling of Flow-Induced Vibration Phenomena
- References

10.1 INTRODUCTION

Even with the advent of very fast digital computers and algorithms for solving the Navier-Stokes equations, engineering fluid mechanics still rely to a large extent on empiricism. For example, the majority of flows of practical interest are turbulent in nature. With the technology of today, the Navier-Stokes equations can actually be directly solved for some turbulent flows (typically, low Reynolds number) using direct numerical simulation (DNS) algorithms. But for most flows of engineering interest, DNS methods are not practical, and solving the Navier-Stokes equations requires an empirically derived turbulence model. Therefore, the point to be made here is that for the foreseeable future, empirical methods will likely continue to hold an important place in engineering fluid mechanics.

A powerful method for analyzing complex flows is scaled physical modeling. In some cases, scaled physical modeling can be conducted more quickly and cost effectively than computational fluid dynamics (CFD) modeling. Often, scaled physical modeling and CFD modeling can be used to complement one another, so that by using them together, problems can be solved that would be either too difficult or too costly to solve using either physical or CFD modeling alone.

Scaled physical modeling relies on the techniques of dimensional analysis, which predate numerical methods for solving the Navier-Stokes equations by at least 50 years. The primary benefit of using dimensional analysis is that of reducing the number of experiments required to characterize a flow phenomenon using physical modeling. The following example illustrates the power of dimensional analysis to reduce what could be a formidable task into a manageable one.

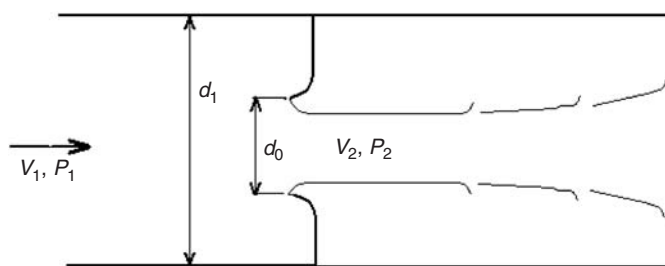


FIGURE 10.1 Flow through an inverted nozzle.

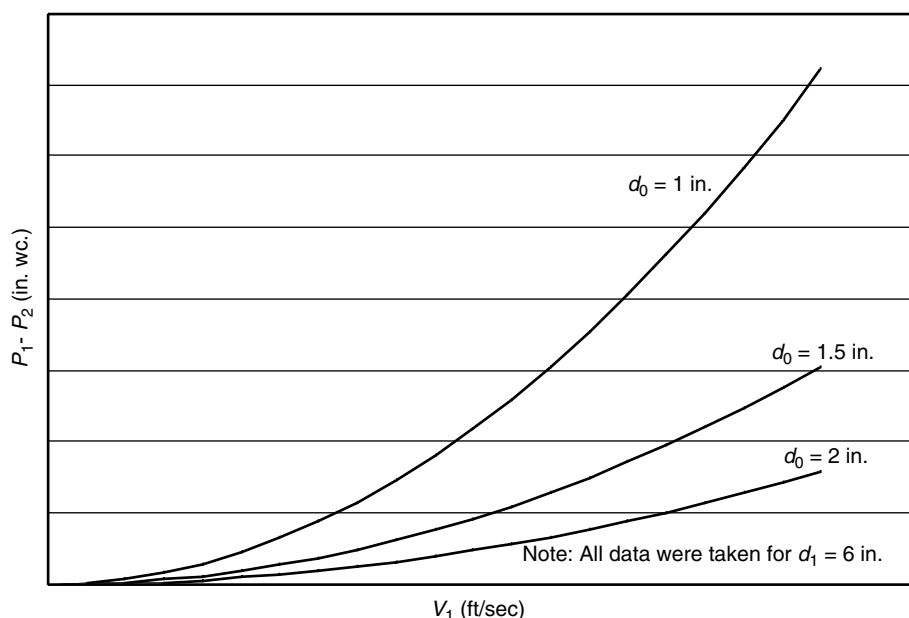


FIGURE 10.2a Pressure-velocity-diameter relationship.

To characterize the static pressure drop ($P_1 - P_2$) through an inverted nozzle, such as shown in Figure 10.1, a brute-force approach would be to conduct a large number of experiments varying d_0 , d_1 , and V_1 , while measuring P_1 and P_2 for each experiment. The result would be developing numerous curves as shown in Figure 10.2a, where each curve represents a large number of tests for fixed values of $d_0 d_1$. If d_1 were varied, then the set of curves shown in Figure 10.2a would need to be regenerated. It can easily be seen that to characterize the system for a wide range of the correlating parameters (d_0 , d_1 , V_1 , P_1 , and P_2), a very large number of tests would be required.

A better approach, which would eliminate many experiments while accomplishing the same objective, is to nondimensionalize the governing equation and define a pressure coefficient. The governing equation for flow through the inverted orifice is the energy equation with a loss term, which is equivalent to Bernoulli's equation with an energy loss term:

$$P_1 + \frac{1}{2} \rho V_1^2 = P_2 + \frac{1}{2} \rho V_2^2 + K_L \frac{1}{2} \rho V_1^2 \quad (10.1)$$

The velocity ratio V_2/V_1 is a function of d_1/d_0 , and the loss coefficient K_L is essentially only a function of d_1/d_0 and the Reynolds number ($\rho V_1 d_0/\mu$). A dimensionless pressure coefficient C_p can be defined as:

$$C_p = \frac{P_1 - P_2}{\frac{1}{2} \rho V_1^2} \quad (10.2)$$

Therefore, Equation 10.1 can be expressed as:

$$C_p = \frac{P_1 - P_2}{\frac{1}{2} \rho V_1^2} = f_1\left(\frac{d_1}{d_0}\right) - 1 + f_2\left(\frac{d_1}{d_0}, \frac{\rho V_1 d_0}{\mu}\right) \quad (10.3)$$

where:

- V_1 = velocity of the fluid upstream of nozzle
- ρ = density of the fluid
- μ = dynamic viscosity of the fluid
- d_0, d_1 = dimensions shown in [Figure 10.1](#)

and C_p is seen to be only a function of d_1/d_0 and Reynolds number ($\rho V_1 d_0/\mu$). The dimensionless pressure coefficient (C_p) represents the ratio of pressure forces to inertial forces.

Instead of conducting many experiments varying d_0 , d_1 , and V_1 to develop families of curves, fewer curves can be developed requiring significantly fewer tests. The curve in Figure 10.2b was generated at a constant Reynolds number, and it contains similar information as shown in Figure 10.2a, except that it is expressed in a nondimensional fashion. By nondimensionalizing the problem, the number of correlating parameters can be reduced from five (d_0 , d_1 , V_1 , P_1 , and P_2) to three

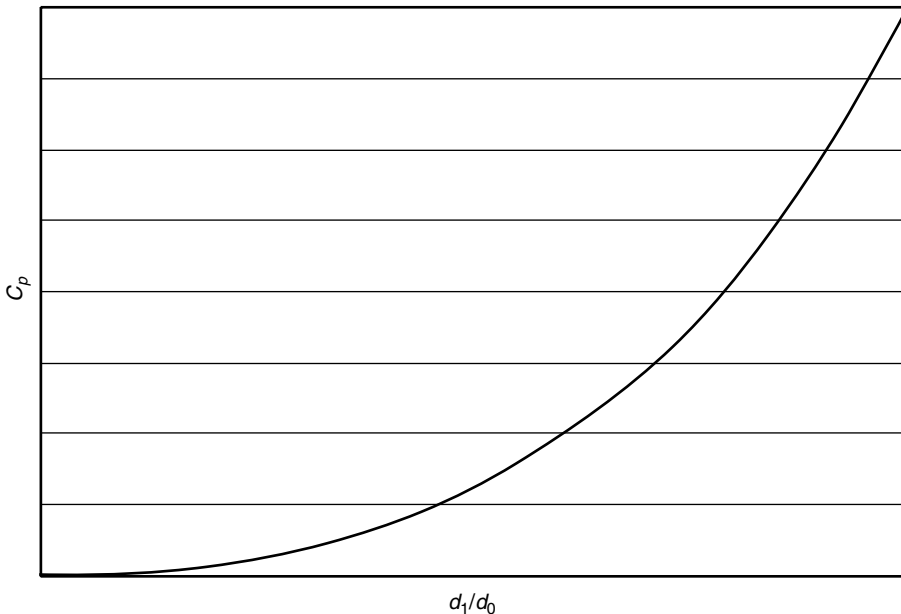


FIGURE 10.2b Nondimensional pressure coefficient.

(C_p , Reynolds number, and d_1/d_0). This example demonstrates the power of dimensional analysis to reduce the number of correlating parameters to the minimum required, and also to significantly reduce the number of tests required to characterize a flow problem. As discussed in the following chapter sections, above a certain Reynolds number, C_p does not vary significantly with Reynolds number for many geometries.

10.2 METHODS OF DIMENSIONAL ANALYSIS

The example of the previous section clearly illustrates the need for using dimensional analysis to conduct flow-modeling studies. There are several good references on dimensional analysis, such as Fox and McDonald¹ and De Nevers.² The Buckingham Pi theorem provides the framework and methodology for determining the pertinent dimensionless parameters for a given flow scenario.

10.2.1 BUCKINGHAM PI THEOREM

For a physical fluid flow problem where a dependent variable is a function of $n - 1$ independent parameters, the relationship between the parameters can be expressed as:

$$q_1 = f(q_2, q_3, q_4, \dots, q_n) \quad (10.4)$$

where, q_1 is the dependent parameter and $q_2, q_3, q_4, \dots, q_n$ are independent parameters. The relationship can be expressed equivalently as:

$$g(q_1, q_2, q_3, q_4, \dots, q_n) = 0 \quad (10.5)$$

The Buckingham Pi theorem says that when a relationship such as Equation 10.5 exists between n parameters, the parameters may be grouped into $n - m$ dimensionless and independent ratios (Π parameters). A set of Π parameters is not independent if any of them can be expressed as a product or quotient of the others.

In addition, the Buckingham Pi theorem states that these $n - m$ dimensionless ratios can be expressed in either of the following functional forms:

$$G_1(\Pi_1, \Pi_2, \Pi_3, \Pi_4, \dots, \Pi_{n-m}) = 0$$

or

$$\Pi_1 = G_2(\Pi_2, \Pi_3, \Pi_4, \dots, \Pi_{n-m})$$

The number m is often, but not always, the minimum number of independent (or primary) dimensions required to specify the dimensions of the $q_1, q_2, q_3, \dots, q_n$ parameters. Examples of independent (or primary) dimensions are mass, length, and time. Force and energy are not primary dimensions because they can be expressed as products and quotients of mass, length, and time. The Buckingham Pi theorem does not state what the function form of the $\Pi_1, \Pi_2, \Pi_3, \Pi_4, \dots, \Pi_{n-m}$ parameters will be. This functional form must be determined through experimentation.

10.2.2 METHOD FOR DETERMINING THE Π GROUPS

The following is a systematic method for determining the Π groups:

Step 1. Develop a list of all pertinent parameters. If it is questionable whether or not a parameter is involved, it is prudent to list it, thereby reducing the likelihood of missing pertinent

parameters. One of the parameters will be the dependent parameter, and it should be identified. Let n be defined as the number of pertinent parameters.

Step 2. Define the primary dimensions (mass, length, time, etc.) for the problem.

Step 3. Express each parameter in terms of its primary dimensions. Let r be defined as the number of primary dimensions required to define the problem.

Step 4. From the list of dependent parameters (**Step 1**), find a number of repeating parameters equal to the number of primary dimensions r . These repeating parameters should include all the primary dimensions from **Step 2**. The repeating parameters should be selected such that they cannot be combined internally into a dimensionless group.

Step 5. Combine the repeating parameters with each of the remaining parameters to form $n - m$ dimensionless groups (Π parameters). Usually, m is equal to the number of primary dimensions r .

Step 6. Verify that each Π parameter is dimensionless.

10.2.3 APPLICATION OF THE BUCKINGHAM PI THEOREM

This section applies the Buckingham Pi theorem to the previously discussed problem for characterizing pressure drop for flow through an inverted nozzle shown in [Figure 10.1](#).

Step 1. Dependent parameter: ΔP , where $\Delta P = P_1 - P_2$

Independent parameters: V, ρ, d_0, d_1, μ ($n = 6$ parameters)

V = velocity of the fluid

ρ = density of the fluid

μ = dynamic viscosity of the fluid

d_0, d_1 = dimensions shown in [Figure 10.1](#)

Step 2. Primary dimensions: M (mass), L (length), t (time) ($r = 3$ dimensions)

Step 3. Expressing each parameter in terms of its primary dimensions:

$$\Delta P: \frac{M}{Lt^2} \quad V: \frac{L}{t} \quad \rho: \frac{M}{L^3} \quad d_0, d_1: L \quad \mu: \frac{M}{Lt}$$

Step 4. Find r repeating parameters: V, ρ , and d_0

Step 5. Combine the repeating parameters with the remaining parameters to form $n - m$, where $m = r$, dimensionless groups or Π parameters.

$$\Pi_1 = (\Delta P) \rho^a V^b d_0^c$$

Substituting primary dimensions for each parameter, $\Pi_1 = \left(\frac{M}{Lt^2}\right) \left(\frac{M}{L^3}\right)^a \left(\frac{L}{t}\right)^b (L)^c$

Step 6. To make Π_1 dimensionless, the exponents for each primary dimension are summed and equated to zero:

$$M: a + 1 = 0$$

$$L: -1 - 3a + b + c = 0$$

$$t: -2 - b = 0$$

Therefore, $a = -1$, $b = -2$, and $c = 0$, and the first Π parameter is shown below:

$$\Pi_1 = \frac{\Delta P}{\rho V^2}$$

Typically, ΔP is made nondimensional by dividing by dynamic pressure; therefore:

$$\Pi_1 = \frac{\Delta P}{\frac{1}{2}\rho V^2}$$

A similar process is conducted for the remaining Π parameters,

$$\Pi_2 = \mu \rho^a V^b d_0^c \quad \text{and} \quad \Pi_3 = d_1 \rho^a V^b d_0^c$$

The remaining Π parameters become:

$$\Pi_2 = \frac{\rho V d_0}{\mu} \quad \text{and} \quad \Pi_3 = \frac{d_1}{d_0}$$

The Π_1 parameter is the pressure coefficient (C_p), Π_2 is the Reynolds number, and Π_3 is the diameter ratio for the orifice. Therefore, $C_p = F(\text{Re No.}, \frac{d_1}{d_0})$, where the functional relationship must be determined through experimentation.

10.3 DIMENSIONAL ANALYSIS AND MODEL STUDIES

When performing a scaled model study of fluid flow phenomena, the model test should yield useful information on forces, pressures, and velocity distributions that would exist in the full-scale prototype. For the model to produce useful information on forces, pressures, and velocities, it must be similar to the prototype in certain respects. There are three similarity types that can be required of a model with respect to the prototype: geometric similarity, kinematic similarity, and dynamic similarity.

Geometric similarity requires that the model and prototype be of the same shape. In addition, the ratio of model dimension to prototype dimension should be a constant scale factor throughout the geometry.

Kinematic similarity requires that velocities be in the same direction at all points in the model and prototype. An additional requirement is that the ratio of model velocity to prototype velocity be a constant scale factor for all locations in the geometry, so that flows that are kinematically similar have similar streamline patterns. Because boundaries of the geometry determine the bounding streamlines, flows that are kinematically similar must also be geometrically similar.

The third type of similarity is dynamic similarity, which exists when model and prototype force distributions are in the same direction (i.e., parallel) and the ratio of the magnitudes is a constant scale factor. Dynamic similarity requires that both geometric and kinematic similarity exist between the model and prototype. To achieve dynamic similarity, all forces (pressure, viscous, surface tension, etc.) acting on the fluid must be considered. The dimensionless groups determined by applying the Buckingham Pi theorem can be shown to be ratios of forces (e.g., the Reynolds number is the ratio of the inertia force to the viscous force). This implies that to achieve dynamic similarity, all of the pertinent dimensionless groups must be matched between the model and the prototype (For example, $\text{Re No.}_{\text{model}} = \text{Re No.}_{\text{prototype}}$). As discussed in a subsequent section, in practicality for many model studies, complete dynamic similarity cannot be achieved but it is still possible to use the model test results to accurately predict forces and pressures in the prototype.

10.4 AIRFLOW REQUIREMENTS FOR PROPER BURNER OPERATION

Up to this point, the need for dimensional analysis and the associated methods and theory have been discussed in some detail. Additionally, Chapter 10.3 discussed how dimensionless analysis is applied in conducting physical flow modeling studies. This section discusses the application of flow model studies for burner design.

There are three very important factors related to airflow for burner designers:

1. For multiple burner units, each burner should receive an equal amount of airflow. For units with FGR (flue gas recirculation), this implies that each burner should also receive the same amount of FGR.
2. The velocity distribution entering each burner should be uniform around the periphery of the air inlet.
3. The airflow in the burner should not have swirl caused by the conditions of the air as it enters the burner. Most burners, by design, generate swirl in the air after it has entered the burner, but the airflow inlet conditions should not generate swirl.

As a result of many observations of multiple-burner, oil and gas firing equipment, on a wide range of boiler designs, it has been concluded that the proper airflow distribution to each burner is essential in order to control flame shape, flame length, excess air level, and overall combustion efficiency. Proper airflow distribution consists of equal combustion airflow between burners, uniform peripheral velocity distributions at the burner inlets, and the elimination of tangential velocities within each burner. If the unit has been designed with windbox FGR, the O_2 content must be equal between the burners, and this is accomplished by balancing the FGR distribution to each burner. Considering that air in the combustion process accounts for approximately 94% of the mass flow, numerous observations on boiler combustion systems have shown that correct airflow distribution is a key factor in the achievement of high performance (low NO_x , low O_2 , and low CO). The concept of equal stoichiometry at each burner results in the minimum O_2 , NO_x , and CO. The most direct way to achieve this is to ensure equal distribution of air and fuel to each burner. Airflow distribution is difficult because it requires a reliable and repeatable flow measuring system in each burner, and a means to correct the airflow without disrupting the peripheral inlet distribution or adding swirl to the airflow.

The remainder of this section explains how each of the three airflow factors relates to a specific burner performance parameter. To achieve the lowest emissions of NO_x , CO, opacity, and particulate, at the minimum excess O_2 , equalization of the mass flow of air to each burner is required. Mass flow deviations should be minimized to enable lower post-combustion O_2 , CO, and NO_x concentrations. The lowest post-combustion O_2 concentration possible is constrained by the burner most starved for air. This starved burner will generate a high CO concentration and, consequently, the total O_2 must be raised to minimize the formation of CO in that burner. By equalizing the airflow to each burner and ensuring that the fuel flow is equal, the O_2 can be lowered until the CO starts to increase equally for all burners. Lower O_2 not only lowers NO_x formation, but also results in higher thermal efficiency. The goal is to reduce the mass flow differences between burners (in the model) to within $\pm 2\%$ of mean. Obviously, this goal becomes inconsequential if the boiler has only one burner.

Flame stability is a very important aspect of the burner and one that appeals to the boiler operator. Flame stability is enhanced in the model by controlling two parameters: peripheral distribution of airflow at the inlet and the inlet swirl number. Flame stability is primarily controlled in the burner design but must be supported by proper inlet conditions. The equalization of the peripheral air velocity at the burner inlet will result in equal mass flow of air around and through the periphery of the flame stabilizer. The flame stabilizer will tend to equalize any remaining flow deviations because of the high velocity developed in this region of the burner throat. The result of

an equal air mass flow distribution through and around the flame stabilizer will be a fully developed and balanced vortex at the center of the outlet of the flame stabilizer. Flame stability and turndown of the burner depend on the condition of this vortex and its attachment to the flame stabilizer. Nonuniform peripheral inlet velocities result in an asymmetrical vortex, which can lead to a multiplicity of problems. An asymmetrical vortex behind the flame stabilizer can lead to a flame that has poor combustion performance and is more sensitive to operating conditions: turndown might be limited, combustion-induced vibrations might be experienced, FGR might cause flame instability at lower loads, light-off by the ignitor might be more difficult, and flame scanning might show increased sensitivity. Another consideration is that low-NO_x burners rely on injection of fuel at precise locations within the burner airflow, and thus it is imperative that the proper airflow be present at the injection locations. A goal in the model is to reduce peripheral airflow deviations to $\pm 10\%$ at each burner entrance.

Swirl number is an indication of the rotational flow entering the burner. The creation of swirling air is a fundamental requirement of all burners. Louvered burners create this swirl by rotating the entire air mass. Unfortunately, this creates a problem at high turndown rates. At low loads (e.g., 10% of full load heat input), excess O₂ is typically 11 to 13%. By swirling the entire air mass, the fuel is diluted to the point where flame stability becomes marginal. Swirling air entering louvered burners (not created by the burner louvers) can result in different burner-to-burner register settings to obtain uniform swirl intensity at each burner. The differing register positions consequently affect the air mass flow at each burner.

An axial flow burner operates on the principle of providing axial airflow through the burner and developing a controlled swirling vortex of primary air at the face of the smaller, centrally located flame stabilizer (or swirler). This concept maintains a stable flame at the core of the burner by limiting dilution at high turndown rates. The secondary air that passes outside the flame stabilizer, however, is most effective if it is not swirling (which is the concept behind “axial flow” burners). Swirling secondary air increases the dilution of the fuel and limits turndown. A goal in the model, for both the louvered burner and the axial flow burner, is to eliminate any tangential velocities entering the burner. The only swirl present must be that created by the burner itself.

The thermal NO_x from a burner increases exponentially with an increase in flame temperature. The introduction of FGR into the combustion air increases the overall mass of the reactants, and hence the products, in the combustion process. The increased mass, as well as the increased reactant diffusion time requirement, reduce the overall flame temperature. The burner with the least amount of FGR will theoretically have the highest flame temperature and will therefore have the highest NO_x. Likewise, the burner with the highest amount of FGR will theoretically have the lowest flame temperature and lowest NO_x. However, due to the exponential nature of the NO_x–temperature relationship, given an equal FGR deviation (e.g., $\pm 10\%$) between two burners, the higher NO_x values from the low FGR burner will outweigh the lower NO_x values from the high FGR burner. Minimizing the FGR deviations will even out the flame temperatures and therefore minimize the overall NO_x formation rate.

To achieve the goals described above, a scaled aerodynamic simulation model, similar to the one shown in [Figure 10.3](#), can be constructed and tested, based on the physical dimensions and flow rates within the field unit. The model shown in [Figure 10.3](#) is for a 24-burner opposed wall-fired utility unit, equipped with FGR and over-fire air (OFA). A scale model constructed of Plexiglas allows for full visualization of the airflow within the windbox/burner configuration and existing ductworks. The goals of the model are primarily accomplished by installing secondary air duct and windbox baffles. The modeler determines the location of baffles and turning vanes within the combustion air/FGR supply system. An additional goal of the modeling is to have minimal impact on the combustion air/FGR supply system pressure drop. This minimizes the effects on the unit fan performance.

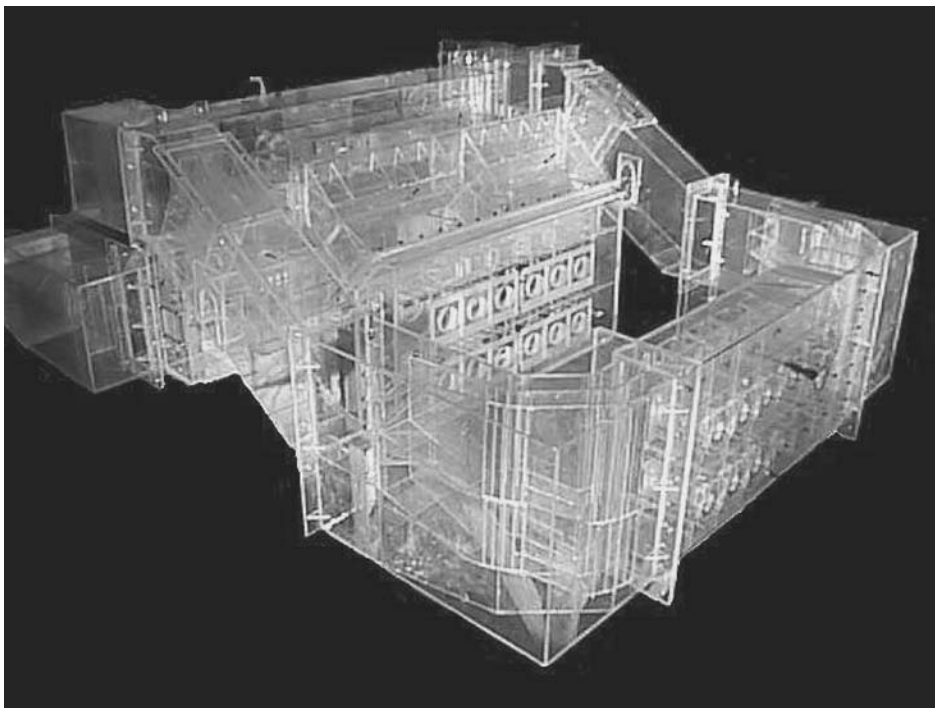


FIGURE 10.3 Windbox model for 24-burner utility boiler.

In summary, the airflow modeling should have four primary goals:

1. To reduce the mass flow differences between burners (in the model) to within $\pm 2\%$ of mean
2. To reduce peripheral airflow deviations to $\pm 10\%$ at each burner entrance
3. To eliminate any tangential velocities entering the burner, so that the only swirl present must be that created by the burner itself
4. To accomplish the first three modeling objectives with minimal impact on overall combustion air/FGR supply system pressure drop

Mass flow deviations from average are shown in [Figure 10.4](#) for a 24-burner utility boiler, and the modeled deviations were within $\pm 2\%$. Also shown in [Figure 10.4](#) are field results on the boiler after correcting the windbox airflow, and mass flow deviations measured in the field were within approximately $\pm 5\%$. A result of meeting the four airflow modeling goals described above is flame-to-flame similarity. [Figure 10.5](#) shows modeled peripheral velocity distribution results for a boiler before and after correction. As seen, the peripheral velocity distribution was very poor before correction. Also shown in [Figure 10.5](#) are flame photographs before correction, and there is a direct correlation between the airflow maldistribution and an unevenly distributed flame shape.

10.5 USING SCALED MODELING FOR ACHIEVING PROPER BURNER AIRFLOW

No modeling work can produce an exact model of reality unless an exact model of the full-scale situation (i.e., another boiler) is made. As discussed in Chapter 10.3, to achieve dynamic similarity, all pertinent dimensionless groups must be matched between the model and the prototype. For airflow modeling of the windbox, connected ductwork, and the burners, geometric and kinematic

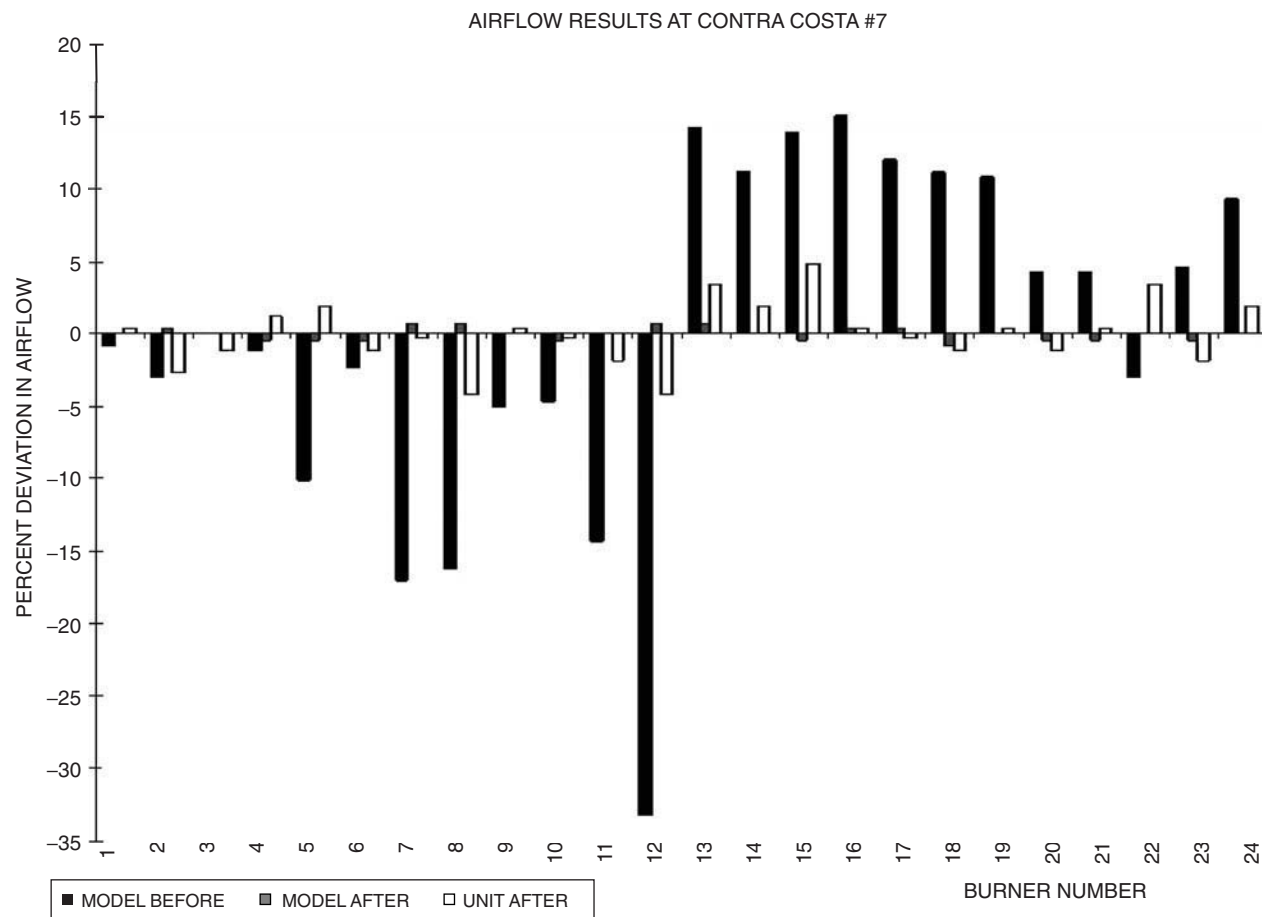


FIGURE 10.4 Improvement of airflow distribution to burners.

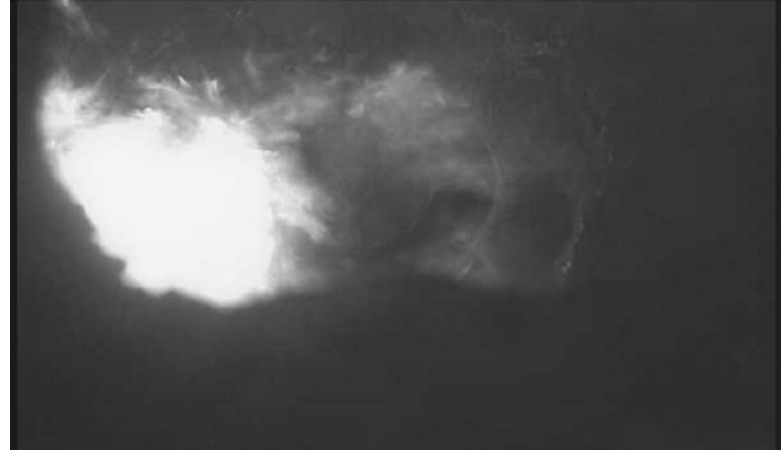
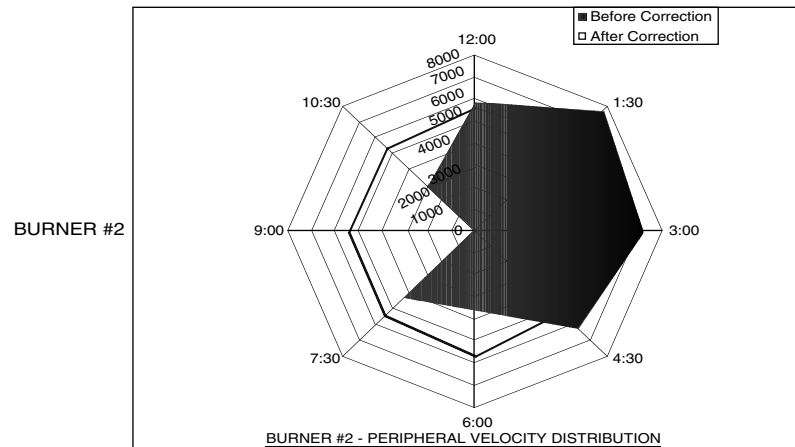
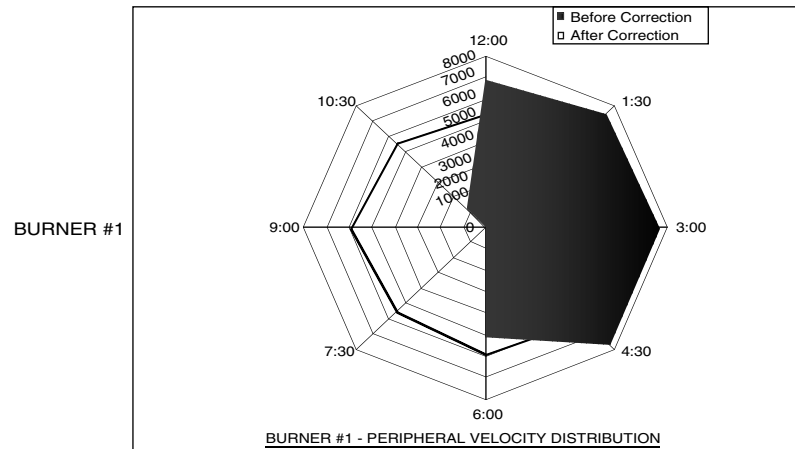


FIGURE 10.5 Effect of peripheral airflow distribution on flame shape.

similitude should be achieved with the scaled model. To achieve dynamic similitude, there are two dimensionless parameters that must be matched: the Reynolds number and the pressure coefficient C_p . This can be verified by applying the Buckingham Pi theorem, which results in Equation 10.6:

$$\text{Re No.}_{\text{model}} = \text{Re No.}_{\text{prototype}} \quad \text{and} \quad C_{p \text{ model}} = C_{p \text{ prototype}} \quad (10.6)$$

where:

$$\text{Re No.} = \frac{\rho V L}{\mu} \quad \text{and} \quad C_p = \frac{\Delta P}{\frac{1}{2} \rho V^2}$$

Air is typically used as the working fluid in the model and the prototype. The model air temperature is typically around 100°F and the prototype air temperature is typically around 530°F (for preheated air). Equating Reynolds numbers, this yields the following relationship between velocities and geometrical dimensions for the model and prototype:

$$\frac{V_{\text{model}}}{V_{\text{prototype}}} \cong 0.37192 \left(\frac{L_{\text{prototype}}}{L_{\text{model}}} \right) \quad (10.7)$$

Model scale is usually around 1/12 ($L_{\text{prototype}}/L_{\text{model}} = 12$) because of physical size constraints, which suggests that:

$$\frac{V_{\text{model}}}{V_{\text{prototype}}} \cong 4.5$$

Prototype duct velocities are typically around 50 ft/sec; this suggests that the model velocity would be in excess of 200 ft/sec to achieve complete dynamic similitude. An air velocity of 200 ft/sec becomes a bit impractical for Plexiglas model construction, and 200 ft/sec would introduce unwanted compressibility effects into the test results. Compressibility effects would result in the model and prototype no longer being kinematically similar, and would require that pitot tube measurements be corrected. For most, if not all, of the losses encountered in windbox and burner geometries, the loss coefficients are relatively constant above a certain Reynolds number. The airflows for windboxes, burners, and associated ductwork are well into the turbulent regime, and it is reasonable to assume that loss coefficients are relatively constant. Therefore, an approach that has been used successfully on many modeling studies is to operate the model at the same velocity as the prototype. The result is that the pressure coefficient C_p will be the same for the model and prototype, which implies that the pressure losses in the model will match those in the prototype. Therefore, experience has shown that although complete dynamic similarity cannot be achieved, it is still possible to use the model test results to accurately predict forces and pressures in the prototype.

10.6 CHARACTERIZATION OF BURNER SWIRL

As discussed, it is important to eliminate the swirl in the burner caused by swirl in the air entering the burner or by a nonuniform peripheral velocity distribution at the inlet. To characterize burner swirl, a swirl number can be computed based on the ratio of angular momentum to the axial momentum. Beer³ recommends an equation for swirl number as given by:

$$S_n \equiv \frac{\int_0^R (Wr) \rho U 2\pi r dr}{R \int_0^R U \rho U 2\pi r dr} \quad (10.8)$$

where:

W = tangential velocity

U = axial velocity

ρ = fluid density

R = burner radius

S_n = swirl number

Experimental studies have shown that the swirl number, as shown above, is an appropriate similarity criterion for swirling jets, produced by geometrically similar swirl generators. Therefore, the swirl number (S_n) as given by Equation 10.8 can be used to characterize swirl intensity in the burner. As discussed, the swirl number can have a significant effect on the flame behavior. The North American Combustion Handbook⁴ provides the following approximate rules, applicable to gas and oil burners. The change in swirl from one category to the next typically has a significant effect on flame shape.

$S_n \sim 0.3$ (Moderate Swirl)

$S_n > 0.6$ (Considerable Swirl)

$S_n > 1$ (High Swirl)

$S_n > 2$ (Very High Swirl)

10.7 TECHNIQUES FOR BURNER MODELING

The discussion of scaled modeling to this point has focused on simulating airflow within the windbox and through the burner without regard for what happens downstream of the burner. Care must be taken to accurately model the jet leaving a burner where combustion takes place. An abrupt change in density occurs as a result of burning the fuel/air mixture. This change in density significantly affects the jet momentum and its rate of entrainment and, therefore, the shape of the jet, as shown in Figure 10.6. Without combustion, capturing the physics in the scaled model poses a problem. Three techniques will be discussed for modeling the important fluid mechanical characteristics of a combusting jet with a scaled isothermal jet:

1. In the Thring-Newby⁵ method, it is assumed that the momentum of the burnt gases controls the fluid mechanics in the furnace. To achieve this hot gas momentum with an isothermal model at room temperature, the model nozzle is exaggerated.
2. The Zelkowski⁶ method attempts to improve upon the Thring-Newby method by using a nozzle that is not as exaggerated, but is displaced back a certain distance.
3. The Davison⁷ method (or Gauze method) uses a strategically placed wire mesh with a certain resistance to artificially create the correct jet shape. The model nozzle is scaled geometrically.

Based on experimental evidence, the Gauze method tends to produce the most accurate results. Therefore, more discussion will be devoted to this method.

10.7.1 THE THRING-NEWBY METHOD FOR BURNER MODELING

As shown in Figure 10.6, the flame front causes the jet to expand, which increases the jet momentum. The Thring-Newby⁵ method assumes conservation of momentum and attempts to account for this increase in jet momentum by enlarging the nozzle area. Using dimensional analysis, the two important dimensionless groups are:

$$\text{Re No.} = \frac{\rho V D}{\mu} \text{ and } \frac{L^2 \rho_f}{A \rho_0}$$

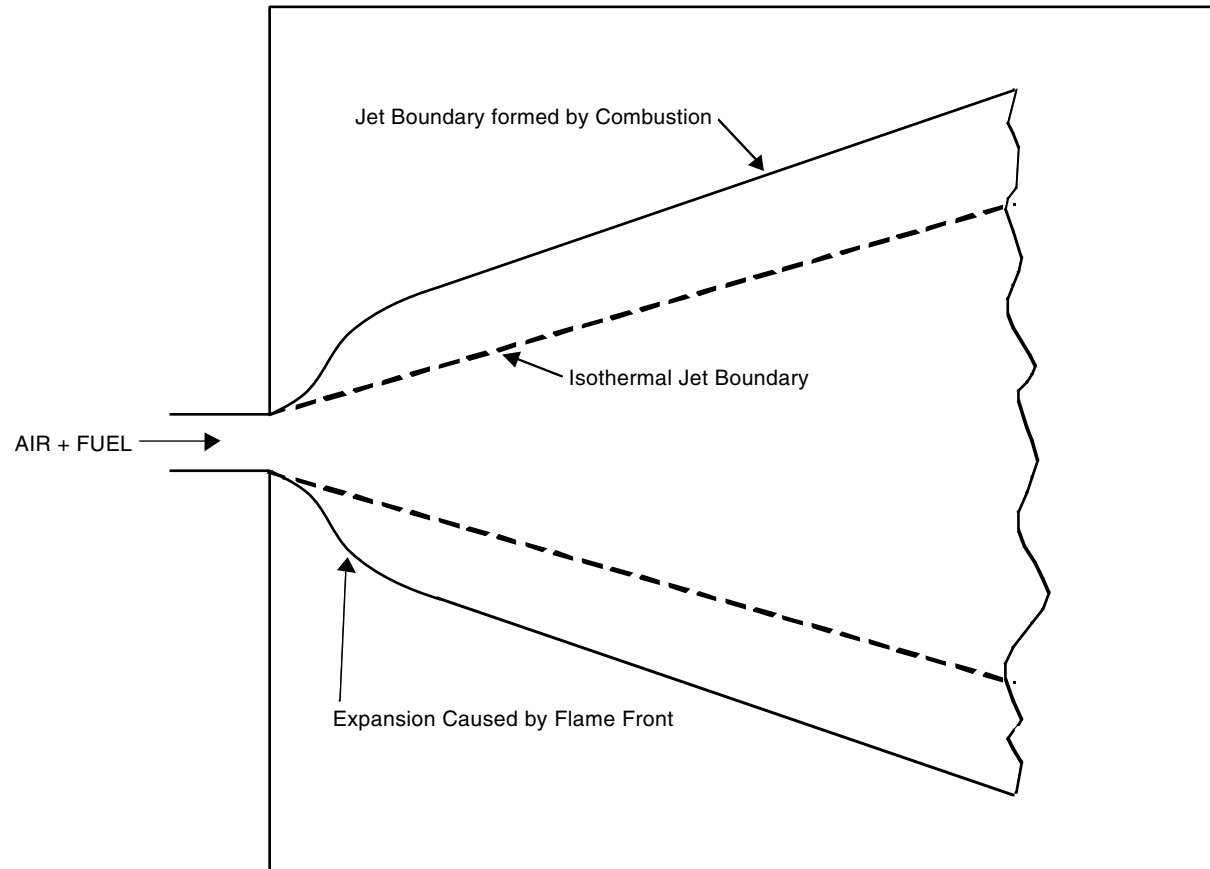


FIGURE 10.6 Combustion jet and isothermal jet boundaries.

where:

L = characteristic dimension

A = model nozzle area

ρ_f = gas jet density at the flame front

ρ_0 = gas density at the nozzle

Anson⁸ indicates that $(\text{Re No.})_{\text{model}}$ can be smaller than $(\text{Re No.})_{\text{prototype}}$ by a factor of 50 with no significant effect on results. Therefore, it is possible to neglect the Reynolds number and only consider the remaining dimensionless group. Neglecting the Reynolds number, the similarity requirement is given by Equation 10.9:

$$\left(\frac{L^2 \rho_f}{A \rho_0} \right)_{\text{model}} = \left(\frac{L^2 \rho_f}{A \rho_0} \right)_{\text{prototype}} \quad (10.9)$$

The model scale factor is defined by:

$$\text{SF} = \frac{L_{\text{prototype}}}{L_{\text{model}}}$$

The working fluid in the model is air; thus, for the model, $\rho_f = \rho_0$. This results in the following relationship for enlarging the nozzle size using the Thring-Newby method:

$$\frac{A_{\text{model}}}{A_{\text{prototype}}} = \frac{1}{(\text{SF})^2} \left(\frac{\rho_0}{\rho_f} \right)_{\text{prototype}} \quad (10.10)$$

The model inlet velocity is typically the same as in the prototype, so that with the enlarged inlet area, the Thring-Newby jet maintains the same momentum of the hot combustion gases at the flame front. Figure 10.7 shows a schematic comparing the Thring-Newby jet with a typical flame shape. The velocity profile at a given cross section is a function of the centerline velocity, which depends on the length of the potential core of the jet. The length of the potential core depends on the nozzle size. Therefore, because of the enlarged nozzle size, the velocity profiles from using the Thring-Newby⁵ method are in error.

10.7.2 THE ZELKOWSKI METHOD FOR BURNER MODELING

Because the nozzle is enlarged with the Thring-Newby method, the length of the potential core is considerably longer than for the actual burner jet. This results in an error in the Thring-Newby velocity profile, which has been estimated by Zelkowski.⁶ Figure 10.8 shows the velocity profile error as suggested by Zelkowski, and Equation 10.11 shows how the velocity error can be calculated. Zelkowski developed a new model to minimize the error as shown by Equation 10.11:

$$\text{VELOCITY ERROR} = \frac{\left[\int_0^{A/r_0} \left(\frac{V_c}{V_0} \right) d \left(\frac{X}{r_0} \right) \right]_{\text{model}}}{\left[\int_0^{A/r_0} \left(\frac{V_c}{V_0} \right) d \left(\frac{X}{r_0} \right) \right]_{\text{actual}}} - 1 \quad (10.11)$$

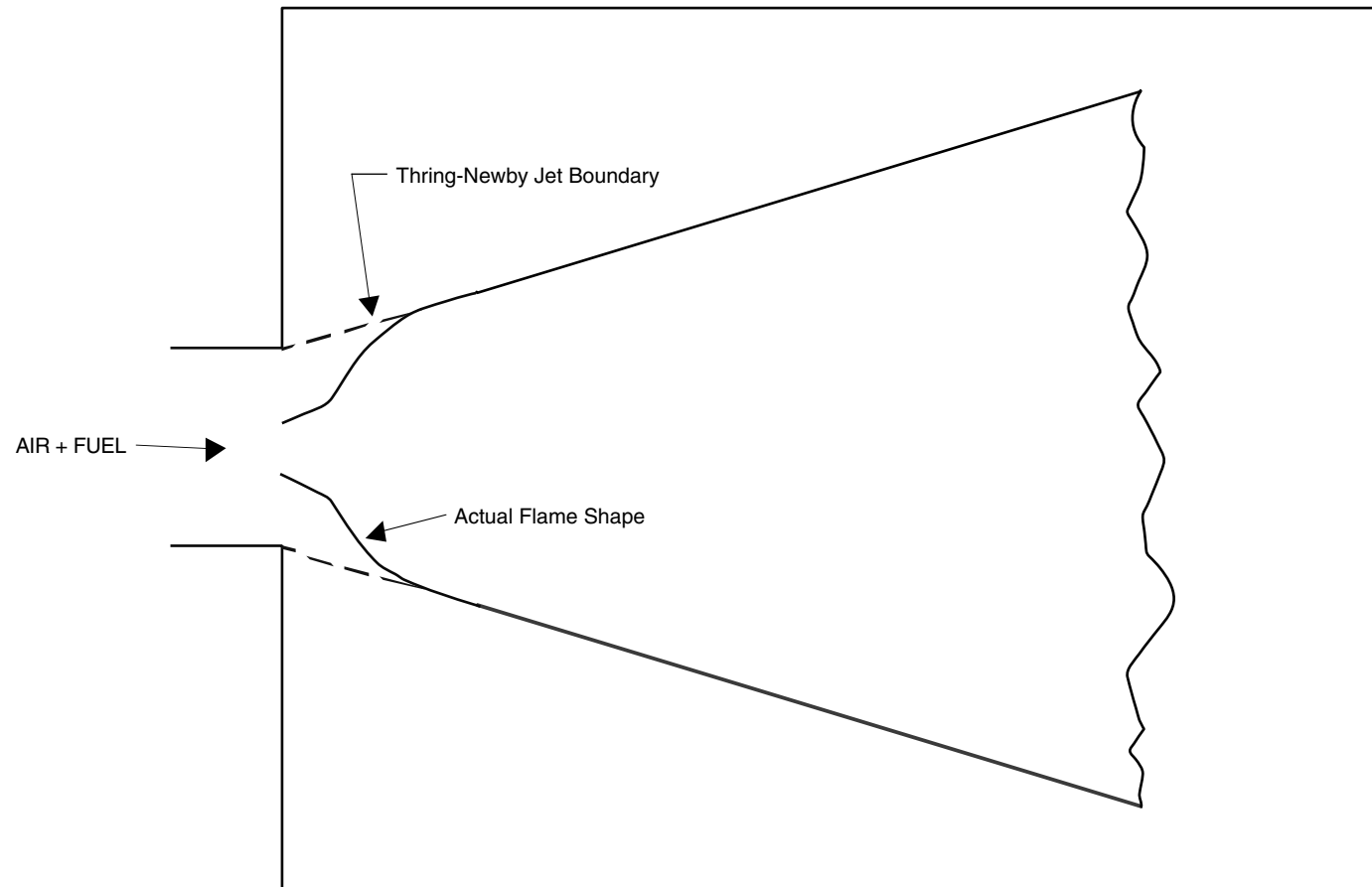


FIGURE 10.7 Thring-Newby jet boundaries.

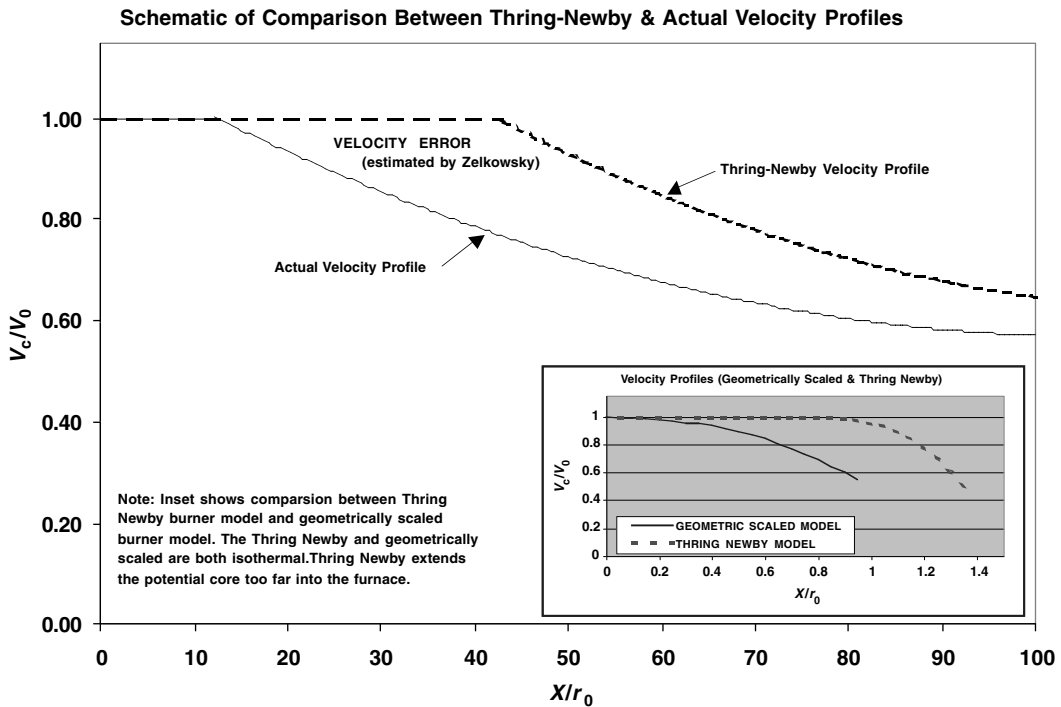


FIGURE 10.8 Thring-Newby velocity errors.

The new model uses a nozzle smaller than that used by Thring-Newby and the nozzle is displaced a certain distance behind the geometrically scaled location. Zelkowsky has put the actual velocity profile, shown in Figure 10.8, and a new parameterized model velocity profile into Equation 10.11, and then found parameters for the new profile that minimize the velocity error (Equation 10.11). Included in the parameters solved for are nozzle size and the distance displaced behind the actual location.

10.7.3 THE DAVIDSON (GAUZE) METHOD FOR BURNER MODELING

The Davison method (or Gauze method)⁷ uses a strategically placed wire mesh with a certain resistance to artificially create the correct jet shape. In the following discussion, two limiting cases are discussed. The first case is a confined jet, and the second case is an effectively free jet, as shown in Figure 10.9 and Figure 10.10, respectively. The wire grid causes the jet to expand which simulates the expansion created by the flame or combustion front. As stated above, in the Gauze method, the model nozzle is scaled geometrically. The authors recommend maintaining the same nozzle velocity in the model and prototype, as shown in Equation 10.12:

$$V_{0,model} = V_{0,prototype} \quad (10.12)$$

The primary reason for this is to prevent significant compressibility effects from occurring. Davison seems to recommend scaling the prototype nozzle velocity ($V_{0,prototype}$) by the density ratio (ρ_0/ρ_f). A typical value of $V_{0,prototype}$ is 100 to 150 ft/sec; therefore, $V_{0,model}$ would easily be around 500 ft/sec, which would introduce significant compressibility effects.

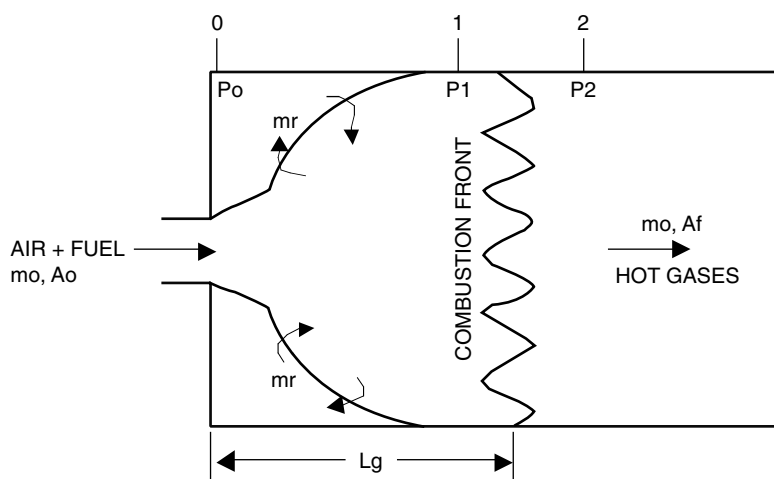


FIGURE 10.9 Confined combustion jet.

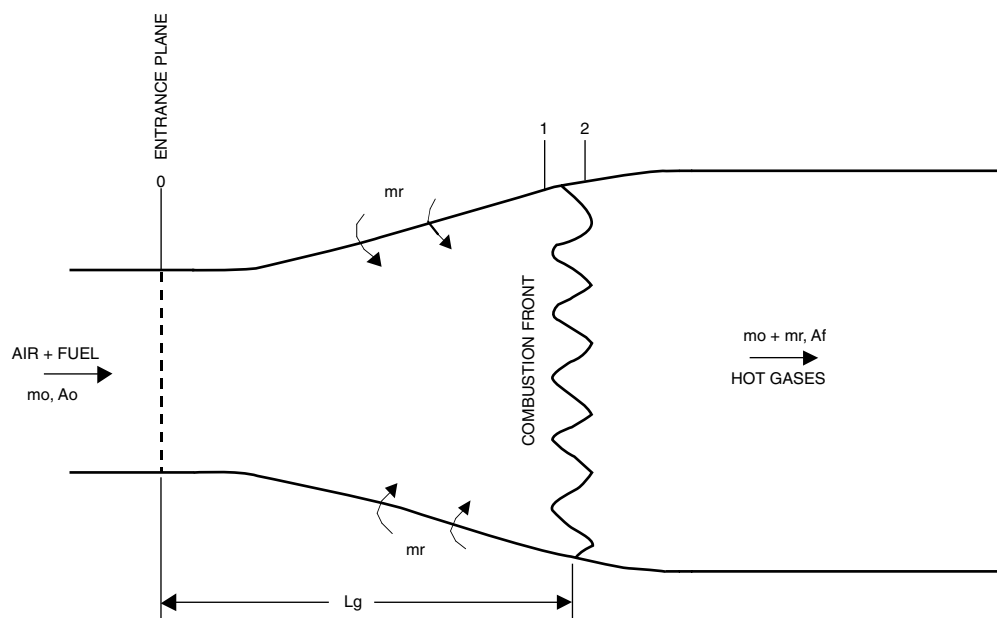


FIGURE 10.10 Free combustion jet.

10.7.3.1 Case A: Free Jet

In the free jet (Figure 10.10), the static pressures at locations 0, 1, and 2 are all equal. The jet is isothermal for the distance (L_g) from the entrance plane to the combustion front. The fluid momentum within the isothermal region entrains surrounding fluid into the jet boundary. The flame causes the gases to expand, which creates a drag force within the jet. The effect of the expansion is similar to placing an obstruction in the jet flow field, as shown in Figure 10.11. If the momentum equation is applied to a control volume around the obstruction, the drag force exerted on the fluid is given

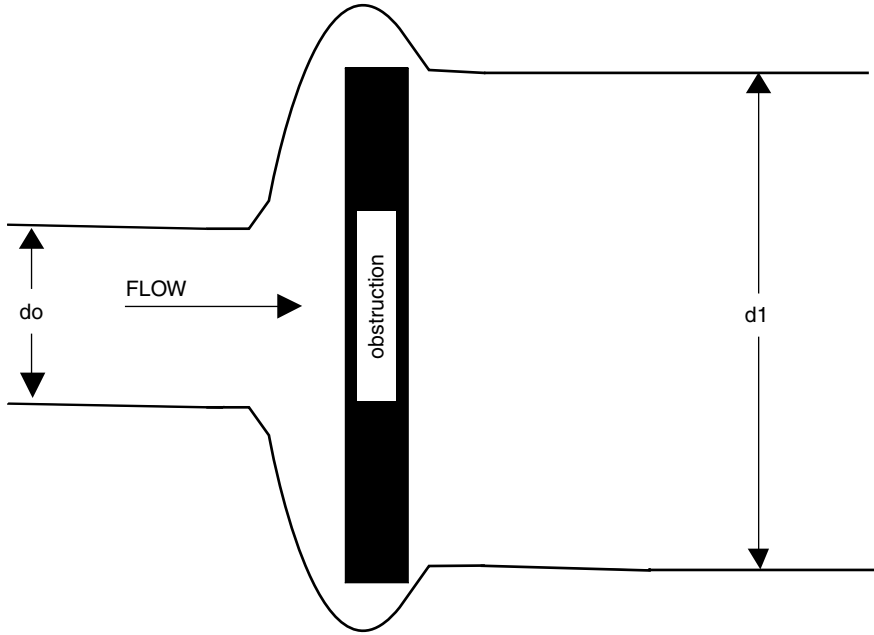


FIGURE 10.11 Jet flow field around an obstruction.

by Equation 10.13, which shows that the drag force on the fluid is a function of the flow areas upstream and downstream of the obstruction.

$$\frac{D}{G_0} = 1 - \frac{A_0}{A_1} = 1 - \frac{d_0^2}{d_1^2} \quad (10.13)$$

where:

D = drag force exerted on fluid by obstruction

G_0 = upstream momentum flux (product of mass flow rate and velocity)

A_0, A_1 = flow areas upstream and downstream of obstruction, respectively

d_0, d_1 = diameters upstream and downstream of obstruction, respectively

As stated above, for the free jet case, as depicted in [Figure 10.10](#), the pressures at planes 0, 1, and 2 are equal. Therefore, for the free jet, the flow area at plane 2 is inversely proportional to the density, $\rho_2 = \rho_f \sim 1/A_2$. Using this result in Equation 10.13 provides a relationship for the drag force of the wire grid, as shown in Equation 10.14:

$$\frac{D}{G_0} = 1 - \frac{\rho_f}{\rho_0} \quad (10.14)$$

where:

ρ_0 = density of nozzle gas in prototype

ρ_f = density of hot gases downstream of the combustion front in prototype

At this point, an analysis will be applied to the model with the wire grid in place. An assumption is made that the entrained flow be neglected. Referring to [Figure 10.12](#), the jet boundaries form

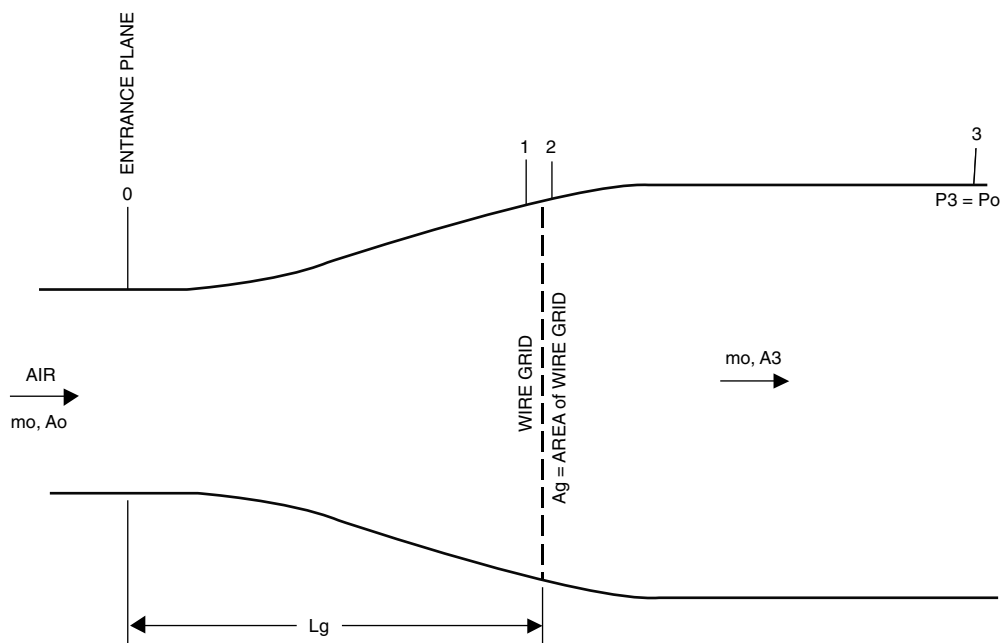


FIGURE 10.12 Modeled isothermal free jet with wire grid.

stream tubes and Bernoulli's equation can be applied between planes 0 and 1, and then between planes 2 and 3. This is shown in Equations 10.15 and 10.16:

$$P_1 - P_0 = \frac{m_0^2}{2\rho_0} \left(\frac{1}{A_0^2} - \frac{1}{A_g^2} \right) \quad (10.15)$$

$$P_3 - P_2 = \frac{m_0^2}{2\rho_0} \left(\frac{1}{A_g^2} - \frac{1}{A_3^2} \right) \quad (10.16)$$

Applying the momentum equation to a control volume bounded by planes 0 and 3, and recalling that $P_0 = P_3$, yields Equation 10.17:

$$\frac{m_0^2}{\rho_0 A_3} - \frac{m_0^2}{\rho_0 A_0} = -D \quad (10.17)$$

where D is the drag force caused by wire grid.

If the momentum equation is applied to a control volume just surrounding the wire grid, bounded by planes 1 and 2, Equation 10.18 is obtained:

$$P_1 - P_2 = \frac{D}{A_g} \quad (10.18)$$

If a resistance coefficient K is defined for the wire grid, the loss across the grid can be expressed by Equation 10.19:

$$P_1 - P_2 = \frac{K}{2} \frac{m_0^2}{\rho_0 A_g^2} \quad (10.19)$$

Combining Equations 10.14 through 10.19 results in a relationship for the wire grid resistance coefficient K in terms of the density ratio, ρ_f/ρ_0 , shown in Equation 10.20:

$$\frac{K}{2} = \frac{2 \left(1 - \frac{\rho_f}{\rho_0} \right)}{\left(1 + \frac{\rho_f}{\rho_0} \right)} \quad (10.20)$$

Davison showed that Equation 10.20 is valid by demonstrating favorable comparisons with experimental data. Therefore, the above assumption of neglecting entrained mass flow in the analysis is acceptable.

10.7.3.2 Case B: Confined Jet

For this case, the jet is constrained by its surroundings, and the pressures at planes 0, 1, and 2 are not equal. Referring to [Figure 10.9](#), the net mass entrainment (m_e) is zero because the jet is constrained by its walls. By applying the momentum equation between planes 1 and 2, a relationship for the pressure drop across the combustion front is obtained. This relationship is given by Equation 10.21:

$$P_1 - P_2 = \frac{m_0^2}{\rho_0 A_f^2} \left(\frac{\rho_0}{\rho_f} - 1 \right) \quad (10.21)$$

The pressure drop across the wire grid in the model can also be related to a resistance coefficient, as given by Equation 10.22:

$$P_1 - P_2 = \frac{K}{2} \frac{m_0^2}{\rho_0 A_f^2} \quad (10.22)$$

Equating Equations 10.21 and 10.22 gives a relationship for the wire grid resistance coefficient:

$$\frac{K}{2} = \frac{\rho_0}{\rho_f} - 1 \quad (10.23)$$

The results in Equations 10.20 and 10.23 assume that the nozzle density in the prototype and the model are the same. If they are significantly different, then Equations 10.20 and 10.23 should be corrected accordingly.

It should be noted that the two cases considered—the free jet and the confined jet—are limiting cases. Many practical cases will likely lie somewhere between these two limiting cases. For example, in the case of a large furnace with multiple burners, adjacent burners will impose some confinement, but there will also be free jet expansion. [Figure 10.13](#) shows a comparison between the resistance coefficients for the two limiting cases discussed above. [Figure 10.13](#) shows that only for small density ratios (ρ_0/ρ_f), the wire grid resistance coefficients for the two limiting cases are reasonably close to one another.

The authors are aware of at least one experimental investigation comparing the Thring-Newby,⁵ Zelkowsky,⁶ and Davison⁷ (Gauze) methods. Based on this study, it was concluded that the Gauze model data was in good agreement with the Desbois⁹ model of vortex flow phenomena, applied to corner fired furnaces. Additionally, the Gauze model data agreed reasonably well with visualization photographs of flow in the lower furnace regions of a prototype furnace. The Gauze method appears to produce more realistic results than the Thring-Newby and Zelkowsky methods.

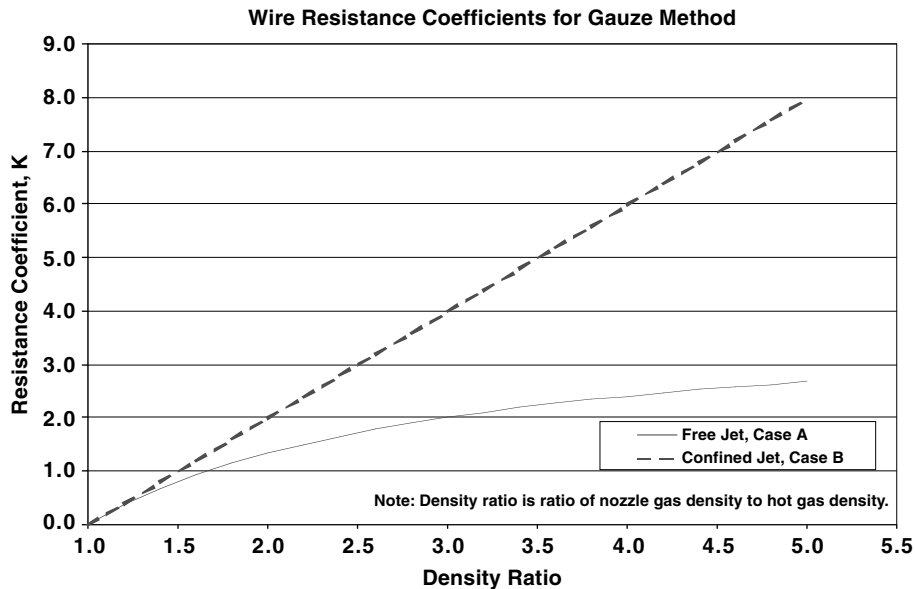


FIGURE 10.13 Gauze method resistance coefficients.

10.8 SCALED MODELING OF FLOW-INDUCED VIBRATION PHENOMENA

An in-depth discussion of flow-induced vibration is beyond the scope of this chapter; however, scaled modeling can be used for diagnosing a type of flow-induced vibration problem. Therefore, a brief discussion is given on this type of modeling. Flow-induced vibration problems can occur when either the structure is excited at one or more of its natural frequencies or when a volume of gas is excited at one or more of its resonance or natural acoustic frequencies. This section discusses flow modeling of flow-induced vibration when the natural acoustic frequencies are excited by vortex shedding.

Most steady flows (constant free stream velocity) over a bluff body, like a cylinder, typically shed vortices at some periodic frequency in the wake of the object. An exception is if the flow does not separate as it passes over the object. For example, for flow over a smooth circular cylinder, periodic vortices are not shed for a Reynolds number below about 40. But for most practical flows dealing with gases, periodic vortices will be shed. The vortices result in pressure fluctuations in the flow field. The nondimensional parameter that governs the shedding frequency is the Strouhal number S_t given by Equation 10.24:

$$S_t = f_s D / U \quad (10.24)$$

where:

f_s = frequency of periodic vortex shedding

D = diameter of bluff body member

U = steady free stream velocity

The Strouhal number varies somewhat with the Reynolds number and depends on the shape of the object. Blevins¹⁰ provides a summary of data showing the effects of Reynolds number and geometric shape on Strouhal number.

As stated, a volume of gas can be excited at one or more of its natural acoustic frequencies. When this condition occurs, sound waves reflect off duct walls to create a pattern of standing waves, and this condition is referred to as acoustic resonance. Baird¹¹ reported this phenomenon in 1954, occurring in a heat exchanger at the Etiwanda Steam Power Station when the power output reached certain megawatt (MW) levels. Baird reported that “.... The vibration was accompanied by intense sound which could easily be heard in the concrete control room some distance away.” Based on this example and others, the vibration energy and potential for damage is high when acoustic resonance occurs.

Acoustic resonance occurs when the gas is excited at one of its transverse natural acoustic frequencies. Transverse directions are perpendicular to the duct axis and the primary flow direction. The following equations, given by Blevins, can be used to compute the transverse natural acoustic frequencies in rectangular and round ducts.

$$f_{a,j} = \frac{cj}{2b}, \quad j = 1, 2, 3, \dots \text{rectangular volume} \quad (10.25)$$

$$f_{a,j} = \frac{c\lambda_j}{2\pi R}, \quad j = 1, 2, 3, \dots \text{cylindrical volume} \quad (10.26)$$

where:

b = width (or height) of the rectangular duct transverse to the primary flow direction

R = radius of the cylindrical duct

c = speed of sound in gas (at the conditions in the duct)

λ_j = dimensionless frequency parameters

$\lambda_1 = 1.841, \lambda_2 = 3.054$

When one of the natural acoustic frequencies matches the vortex shedding frequency (Equation 10.24), within some tolerance, the probability of acoustic resonance is high.

Investigations by Blevins¹² has shown that scaled modeling can be used to reproduce the acoustic resonance condition. For a geometrically scaled model, the two parameters that must be matched are Mach number and the acoustic reduced velocity. Equations 10.27 and 10.28 define these two parameters:

$$\text{Mach number:} \quad \left(\frac{U}{c} \right)_{\text{model}} = \left(\frac{U}{c} \right)_{\text{prototype}} \quad (10.27)$$

$$\text{Acoustic reduced velocity:} \quad \left(\frac{U}{fL} \right)_{\text{model}} = \left(\frac{U}{fL} \right)_{\text{prototype}} \quad (10.28)$$

where:

U = flow velocity

c = speed of sound in gas

f = acoustic natural frequency

L = characteristic length

REFERENCES

1. R.W. Fox and A.T. McDonald, *Introduction to Fluid Mechanics*, fourth edition, John Wiley & Sons, New York, 1992.
2. N. De Nevers, *Fluid Mechanics*, Addison-Wesley, Reading, MA, 1970.
3. J.M. Beer and N.A. Chigier, *Combustion Aerodynamics*, Krieger Publishing, Malabar, FL, 1983.
4. R.J. Reed, *North American Combustion Handbook*, third edition, Vol. II, Cleveland, OH, 1995.
5. M.W. Thring and M.P. Newby, Combustion Length of Enclosed Turbulent Flames, *Fourth (International) Symposium on Combustion*, The Combustion Institute, 1953.
6. J. Zelkowski, Modelluntersuchungen uber die Stromung in Feuerraumen, *Brennst. – Warme – Kraft* 25, 1973.
7. F.J. Davison, Nozzle scaling in isothermal furnace models, *Journal of the Institute of Fuel*, pp. 470–475, 1968.
8. D. Anson, Modeling experience with large boilers, *Journal of the Institute of Fuel*, 40, 20–25, 1967.
9. G.P. Debois, A Study of the Vortex Flow Phenomenon as Applied to Corner Fired Furnace, Masters' project, Sir George Williams University, Montreal, Quebec, 1970.
10. R.D. Blevins, *Flow-Induced Vibration*, second edition, Krieger Publishing, Malabar, FL, 2001.
11. R.C. Baird, pulsation-induced vibration in utility steam generation units, *Combustion*, 25, 38–44, 1954.
12. R.D. Blevins and M.K. Au-Yang, Flow Induced Vibration with a New Calculations Workshop, Continuing Education Short Course Notes, American Society of Mechanical Engineers, New York, 2002.