Numerical Simulation Models for a Modern Boiler Design

T.V. Mull, Jr.
M.W. Hopkins
D.G. White
Babcock & Wilcox
Barberton, Ohio, U.S.A.

Presented to:
Power-Gen International '96
December 4-6, 1996
Orlando, Florida, U.S.A.

BR-1627

Abstract

Numerical simulation models have become an increasingly important design and analysis tool for boiler components and auxiliary equipment. Mature numerical simulation programs with affordable high speed desktop workstation computers have resulted in an ever expanding list of numerical applications. General capabilities accommodate any combination of fossil fuel combustion (coal, oil, and natural gas), non-reacting flows with energy exchange, or isothermal flow problems. Model complexity can be tailored to specific areas of interest. The goal of these numerical simulations is to improve the design of a new or existing boiler component by optimizing the flow distribution. Burner and port arrangements, flow splitters, turning vanes, and perforated plates can be tested numerically to evaluate their effectiveness for improving the final design.

Examples of boiler related numerical applications include furnaces, air supply ducts, erosion analyses, windboxes, steam drums, coal piping, precipitators, pulverizers, burners, and scrubbers. Design considerations for evaluating numerical models depend on the type of problem analyzed. For example, isothermal non-reacting models typically focus on improving the fluid flow distribution (air, water, etc.) to reduce unbalances. For combustion models, the uniformity of flue gas species is weighted against the resulting carbon monoxide and nitrogen oxides emissions. Models involving particle dispersion examine various flow devices to improve, for example, a coal nozzle peripheral distribution or the ash loading pattern in a precipitator. Numerical models are advantageous over physical flow models and field tests because numerous design arrangements can be analyzed without modifying plant equipment. Costs to perform a numerical study are generally one-half to one-third of the cost to perform a physical flow model and can be one-forth of the cost to perform a field study. This paper discusses (1) results from several typical applications to new, existing, and retrofit equipment, (2) the influence on product design and development with the advent of numerical simulation results, and (3) existing numerical limitations and future numerical capabilities.

Introduction

Conventional engineering analyses rely heavily on empirical correlations and experience to develop boiler and auxiliary equipment designs. Today's design processes must be more accurate while minimizing development costs to compete in a world economy. This forces engineering companies to take advantage of design tools which augment existing experience and empirical data while minimizing cost. One tool which excels under these conditions is numerical modeling.

Numerical modeling is used to simulate the physical flow, heat transfer, and combustion phenomena of solids, liquids, and gases. The simulation is completed with the use of computational fluid dynamics (CFD) software executing on high speed, large memory workstations. Numerical modeling has significant cost advantages when compared to physical modeling and field testing. Also, numerical modeling provides additional insight into the physical phenomena being analyzed due to the availability of data that can be analyzed and the flexibility with which geometric changes can be studied.

The software used for these analyses is an active area of development for numerous companies. General analysis software can be purchased from a number of developers. Specialized software can be contracted for development but is more likely to be developed in-house, incorporating a company's technical experience for validation. Experience has demonstrated that purchased software packages by themselves are not adequate to produce reliable numerical results. Reliable results are achieved when the numerical results are used to augment traditional boiler design and analysis tools and empirical data. Specific software packages will not be discussed within this paper.

Technical Approach

The approach used for numerical simulations varies depending on the type of problem solved. If a common denominator exists in the application of boilers, it is flow uniformity, meaning the goal of most flow related problems is to achieve a uniform flow distribution. This can be interpreted to mean equal flow rates between parallel paths, uniform particle loading, or equal distribution of gas species.

The numerical models discussed here are primarily flow related problems and were solved using finite volume analysis tools. Finite volume analyses divide the solution domain into a continuous region of two- or three-dimensional spaces (control volumes) that describe the problem's geometry. Appropriate boundary conditions are applied and the governing physical equations are then solved using an iterative technique.[1] The resulting solution describes the flow, temperature, and chemistry within the boundary. The equations which are solved are algebraic approximations of differential equations; as such, the solution is said to be approximate. The accuracy of the predictions improves as model refinement increases.

A numerical analysis begins with an understanding of the physical processes involved in a particular problem. This may seem to be a trivial statement but actually is essential to obtaining accurate results. Models are tailored to contain refinement in critical areas of interest. Without a good understanding of the physical processes, the model may not contain sufficient refinement, and the accuracy of the results could be compromised. Refinement can have several meanings. It can mean the fineness or level of grid utilization (the number and location of control volumes), the accuracy used to define boundary conditions, or the depth of user defined chemical reactions and subroutines used by the computer program solving the model.

The solution domain for the problem is typically larger than the explicit area of interest. That is because the model must capture the influence of upstream and downstream flow effects at the area of interest. Models typically start from two major flow obstructions upstream of the area of interest. For ducts, this would translate into two upstream turns or bends. For combustion problems, models typically start at the furnace wall openings and require sufficient detail to describe the burner flow characteristics to obtain accurate results. Models typically end one major flow obstruction downstream of the area of interest.

Numerical analyses typically evaluate parametric cases with varying inlet conditions and/or geometries. This provides an inexpensive method to evaluate the effectiveness of turning vanes, perforated plates, and burner and port arrangements. The disadvantage of this methodology is the difficulty in estimating the number of parametric cases required to achieve a final design. Trial arrangements continue until an acceptable arrangement is found. One modeling assignment may require only two or three evaluation cases while the next assignment requires twenty.

Applications

Numerical modeling is routinely used in the design of air staging systems[2] and secondary air windboxes[3] and continues to expand into additional areas of boiler design. The following sections discuss several recent applications of numerical modeling. These examples are intended to provide some insight to the type of problems applicable to numerical models. They include, but are not limited to: pulverizers, micronized coal nozzles, coal gasification, convection pass erosion, scrubbers, and steam drums.

Pulverizer Application

Coal pulverizers rely on a uniform air flow distribution to entrain the varying sizes of pulverized coal particles. If the primary air flow is poorly distributed as it enters the wear track region, several different problems can occur. One problem is that excess air in some regions of the mill will be able to pick up large coal particles that have not had a chance to be properly pulverized. This can lead to choking of the mill at the classifier, reducing the total mill output. Another more serious problem exists when the coal particles fall back into the lower sections of the mill due to low air velocity regions. The resulting coal buildup can create hazardous conditions such as a mill fire or explosion. Thus, it is important to ensure proper air distribution in the pulverizer.

A numerical analysis was performed on a coal pulverizer that was experiencing high coal reject rates believed to be caused by poor primary air distribution in the pulverizer windbox. A three dimensional isothermal flow model was analyzed from the outlet of the primary air fan through the pulverizer windbox throat. The numerical model is shown in Figure 1 and illustrates the solution domain. The subdivision of the geometry into individual control volumes is well illustrated in this figure.

The results showed high air velocities in the duct entering the pulverizer windbox. These conditions resulted from the physical arrangement of the ductwork and the proximity of the primary air fan. Low velocity air flow regions are created in the pulverizer throat near the mill inlet and are shown in Figure 2. These low velocity regions increase the pulverizer's coal reject rates.

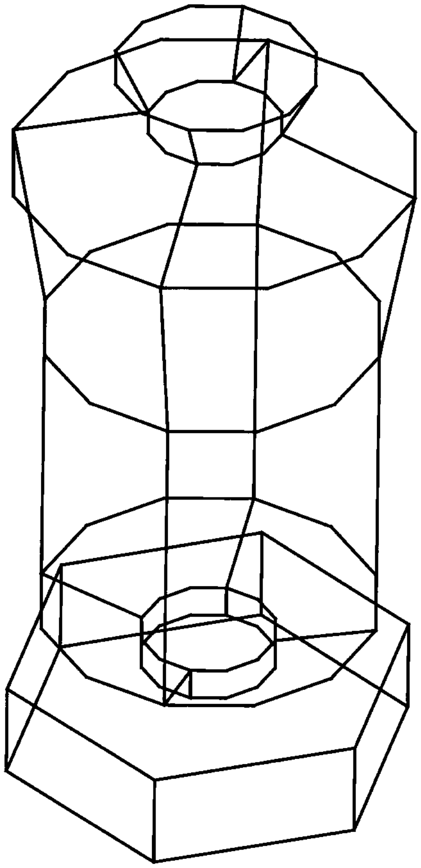

Figure 1 Pulverizer windbox grid.

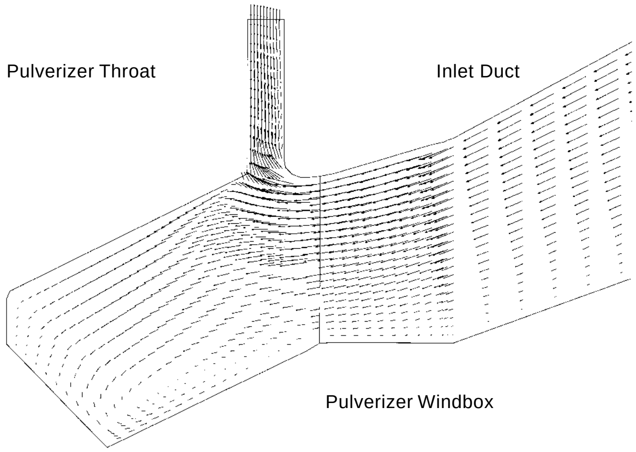

Figure 2 Pulverizer numerical model base case vector plot.

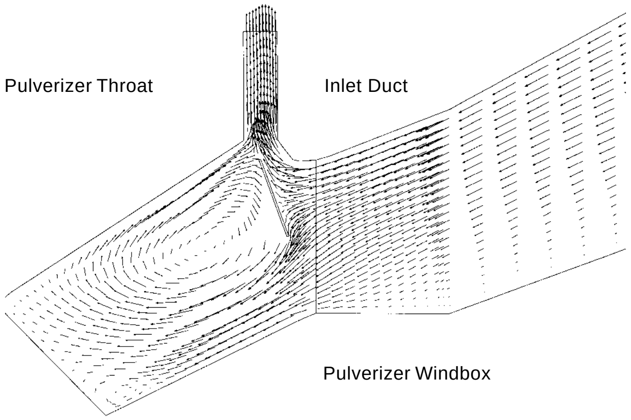

Figure 3 Pulverizer numerical model final case vector plot.

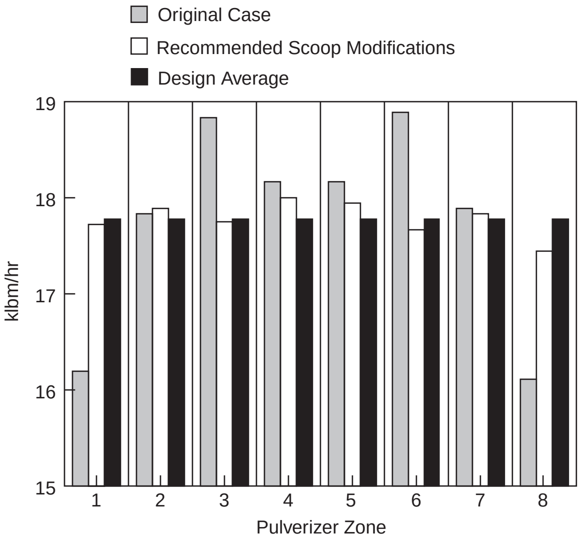

Figure 4 Mass flow in pulverizer zones.

Improvements were designed and installed which included an air scoop in the upper section of the ductwork extending into the mill. Figure 3 illustrates the air scoop and the resulting flow distribution. The scoop redirected air through the pulverizer throat at the entrance region of the mill. The result is an improved total air distribution through the pulverizer throat as seen in the bar chart in Figure 4. The windbox throat was divided into eight equal regions for the purpose of this analysis. The regions were numbered in a clockwise direction starting on the inlet duct centerline. Figure 4 relates the air flow through the eight regions and charts the improvement in total air distribution. The design bar is based on uniform flow to each of the eight regions in the pulverizer windbox.

Micronized Coal Nozzle Application

Some boiler applications utilize a micronized coal which is an order of magnitude smaller than pulverized coal. These applications encounter special problems in the transportation of the particles. The small particles tend to reattach with each other forming coal deposits on inner surfaces which eventually plug the air flow. Numerical modeling is used to determine where the buildups are forming and why. Flow modification devices are then designed to reduce and/or eliminate these buildup regions.

An example of this application was a case utilizing a micronized coal fired burner. The burner was exhibiting a large amount of coal buildup around a turning region inside of the burner, ultimately plugging it. A three dimensional isothermal flow model with Lagrangian coal particle tracking was developed for this application. The numerical model grid is shown in Figure 5.

The results showed that the coal particles were impacting and collecting on the underside of the oil gun and tempering air duct inside the burner. The tempering air duct was removed from the primary air stream while the tempering air was forced along the under side of the oil gun. This allowed the tempering air to buffer the underside of the oil gun helping the primary air and micronized coal to turn along the burner and into the furnace. Figure 6 shows a velocity vector plot for this arrangement with the tempering air duct removed.

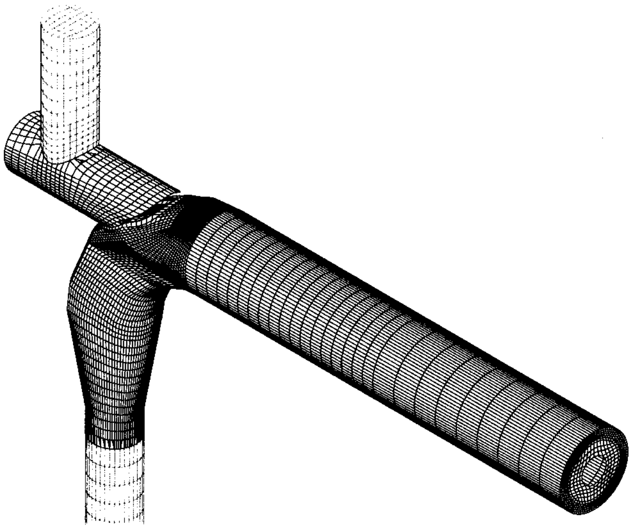

Figure 5 Micronized coal-fired burner grid.

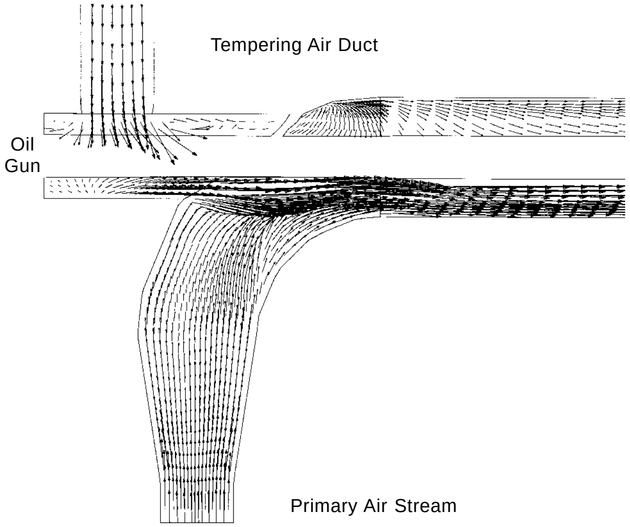

Figure 6 Vector plot; final design.

Coal Gasification Application

A numerical flow and combustion project was completed to study the performance of an entrained flow type coal gasification process. This atmospheric process burns pulverized coal with oxygen in sub-stoichiometric conditions to produce a useful, clean gas. A key design and operational parameter for gasifiers is carbon efficiency. Carbon efficiency represents the fraction of carbon that is converted to a gas phase and establishes the gas composition at the combustor exit. In practice, steam is often injected at the burners in an effort to improve carbon efficiency.

The analysis focussed on the impact to carbon efficiency when varying the oxygen to carbon ratio and when using steam injection at the burners. A common assumption[4] used in the past for designing gasifiers was to predict the combustor exit temperature and base the major gas species on the water gas shift equilibrium equation:

$$CO(g) + H_2O(g) \longleftrightarrow CO_2\,(g) + H_2\,(g)$$

The numerical model showed that, depending on the residence time in a given gasifier, the gas composition at the combustor exit may not be in equilibrium. The predictions of gas composition from the model are determined from chemical kinetics. As such, the rate of progress for a reaction depends on the temperature of the gas and the time the reaction is allowed to proceed. One type of entrained flow gasifier known as the Koppers Totzek design[5] did not reach equilibrium at the combustor exit for every operating condition due to the relatively short residence time of the gas. Consequently, these predictions required designers to rethink some commonly used assumptions.

The analysis also evaluated carbon conversion with varying inlet conditions.A base condition was simulated with a design oxygen to carbon ratio. A lower oxygen to carbon ratio was then simulated with and without steam injection, respectively. Conclusions were that carbon conversion can be predicted based on input oxygen to carbon ratio. Steam injection has little effect when compared to oxygen for evaluating carbon conversion.

The steam injection does however reduce the peak flame temperatures by the endothermic process of the water shift reaction. The reduction in temperature is illustrated in Figure 7.

Figure 7 is a surface plot of constant temperature at 2500^{o}K for one end of the gasifier. The results for this elliptically-shaped gasifier indicate that for steam injection to improve carbon conversion, there must be sufficient oxygen to maintain the higher temperature region. Again, the numerical model provided insight to the combustion phenomena and direction for operating the gasifier with steam injection.

Convection Pass Erosion Application

Erosion plagues numerous areas of power plant equipment. One such example is the convection pass region in boilers firing pulverized coal. In this example, convection pass tubes are eroded when ash particles are conveyed by flue gas and impact the tube surface. The impaction of a particle removes a small amount of tube material. Repeated over long periods of time, the tube can fail as the thickness is no longer adequate to support the required temperature verses pressure stress conditions.

Base Condition

Reduced O_2/C Ratio

Reduced O_2/C Ratio
With Steam Injection

Figure 7 Constant temperature surface plot @ 2500°K for coal gasification.

The particles causing the erosion are primarily ash. The abrasive qualities vary with fuel, however, the erosion rate, E, is defined in open literature[6] as:

$$E = A * V^n$$

Where A is a constant dependent on the particle mineral content, the tube material, and the particle impaction angle; V is the particle speed; and n varies depending on the tube material. Note that for most utility applications, the factor relating impact angle reduces as the flow is channeled past the first few tube rows.

The tube wall material and thickness are typically established by oxidation limits and temperature verses pressure stress considerations. Also, the design fuel establishes the abrasiveness properties of the ash. The only remaining design parameter to minimize erosion is velocity. A reasonable value for n is 2.5 for carbon steel tubes. Therefore, a reduction in gas velocity of 10% results in approximately a 23% reduction in tube erosion.

Numerical models have successfully been used to identify regions of high local velocity and thus erosion rates. This analysis tool can then be utilized to recommend geometric changes or flow modifying devices to reduce the peak velocities, extending the life of tube banks.

One example of an erosion analysis is the following vertical pendant. This superheater experienced high erosion rates at the top of the pendant sections near the penthouse. A two dimensional flow model was used to design a series of baffles along the roof to lower the peak velocities and reduce erosion. Figures 8a and 8b are profile plots of gas velocity entering and leaving the tube bank, without and with baffles, respectively. The velocity profile plots show a reduction in peak velocity of 16 ft/sec which corresponds to a 40% reduction in tube erosion. The final baffle arrangement was determined through numerous parametric models resulting in a series of baffles constructed of 50% open perforated plate hanging vertically from the roof.

Scrubber Application

Coal combustion can cause high SO_2 emissions depending on the sulfur content in the fuel. Wet scrubbers are used to remove sulfur from the flue gas before the gas is released to the atmosphere in a process known as desulfurization.

Flue gas desulfurization takes place in an absorber tower. The flue gas enters at the bottom of the tower and turns immediately upward. The slurry, a controlled mixture of water and limestone, is injected downward into the gas stream through a series of spray nozzles, directed opposite the gas flow, where it absorbs the SO_2. Poor distribution of flue gas inside the scrubber will reduce overall SO_2 removal efficiency. A perforated

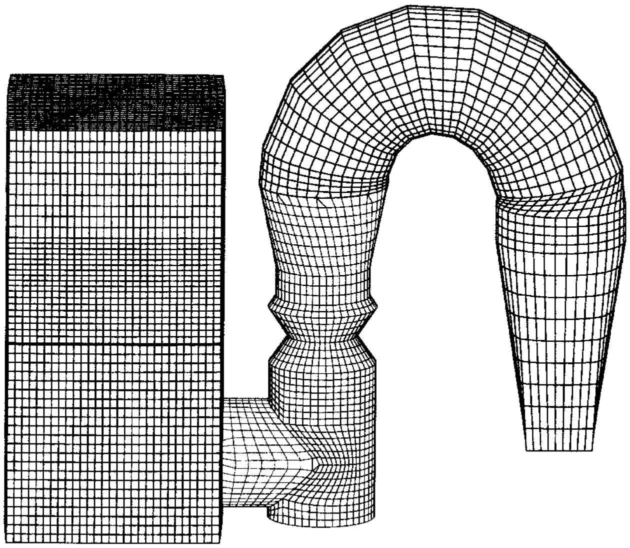

Figure 9 Scrubber grid.

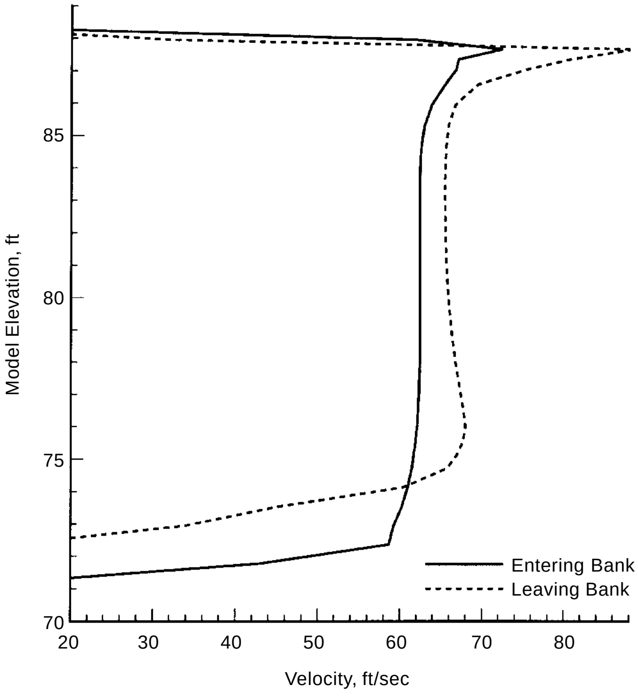

Figure 8a Existing superheater – no baffles.

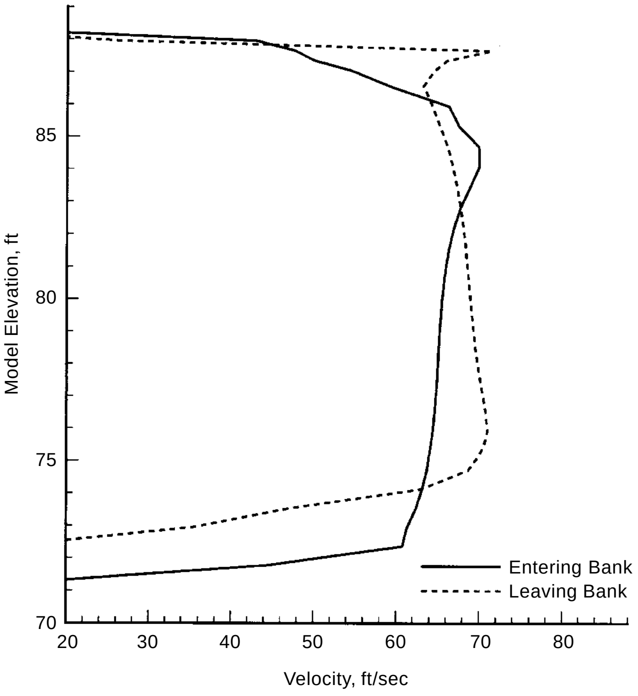

Figure 8b Recommended baffle arrangement.

plate, or tray, is located between the gas inlet and the slurry spray nozzles. The tray acts as a gas-liquid contacting device which allows for additional SO_2 to be absorbed by the slurry. The slurry passes through the tray and drains into the bottom of the tower.

A numerical model was used to design flow modification devices which provide uniform flow at the absorber tray. The model began at the ID fan outlet to account for any effects the upstream geometry had on the flue gas distribution at the scrubber inlet. The numerical grid is shown in Figure 9. The slurry was modeled using Lagrangian particles with a typical size distribution. The normal liquid level inside the absorber tower was represented by a solid surface.

A base case was run with no flow modification devices. The physical arrangement coupled with high gas velocities caused extremely poor gas distribution at the absorber inlet and at the tray. Figure 10 shows a vector plot of the base case. The numerical model indicated that installing turning vanes at the inlet to the absorber greatly improved the gas distribution. The final arrangement also included an inclined plate inside the tower to further improve the gas distribution at the tray. Results from the final arrangement are shown in Figure 11.

Steam Drum Application

The size and number of downcomers on steam drums have a significant impact on unit performance and cost. A design which has not been optimized could contain additional downcomers increasing cost or too few downcomers causing operational problems. Numerical modeling has been used to assist in optimizing the number of downcomers. Critical to this evaluation are water side circulation, feed water thermal mixing, thermal stress, drum water level control, overall cost, maintenance, and construction. Numerical modeling was utilized to analyze the flow characteristics within the steam drum, perform a thermal mixing analysis of the feed water distribution, and evaluate a thermal stress model for a section of the drum head and shell.

The analysis presented here consists of a comparison between a four downcomer and a three downcomer design. The four downcomer design utilized two end and two shell downcomers. The three downcomer design utilized all shell downcomers. The numerical model analyzed the flow distribution of the saturated liquid which provided insight to the potential of carry-over (water flooding the cyclone separators) and carry under (entrained steam entering the downcomer). Shell temperature differentials were examined to minimize thermal stress on the pressure vessel extending the useful life of the steam drum.

Extensive detail was included in the numerical model for accuracy. The model incorporated conjugate heat transfer to represent the feed water and saturated liquid thermal mixing. Steam drum internals were modeled and included the cyclone separators, cyclone directional spouts, the downcomer itself for a length of five diameters, and the downcomer vortex inhibitor.

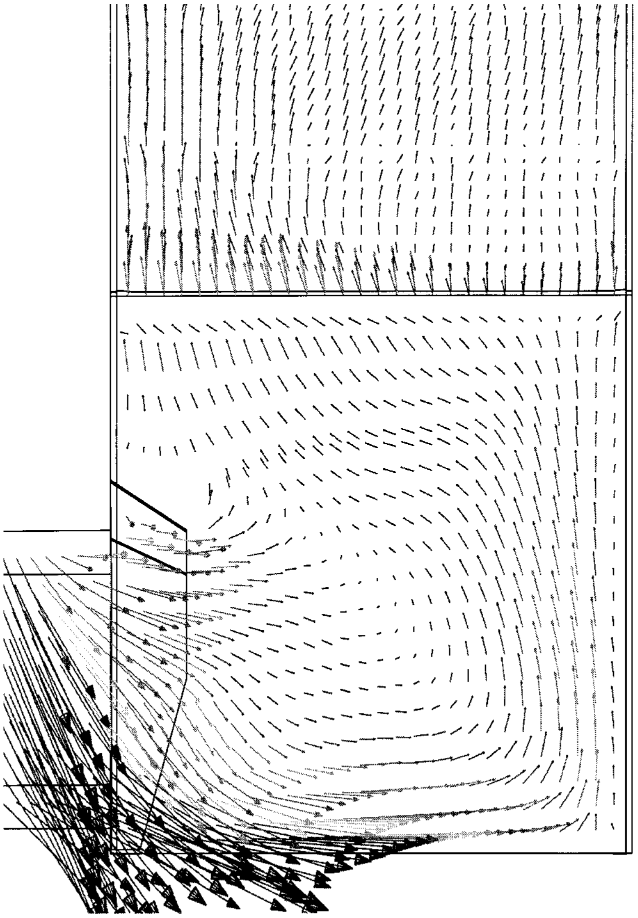

Figure 10 Vector plot; base case.

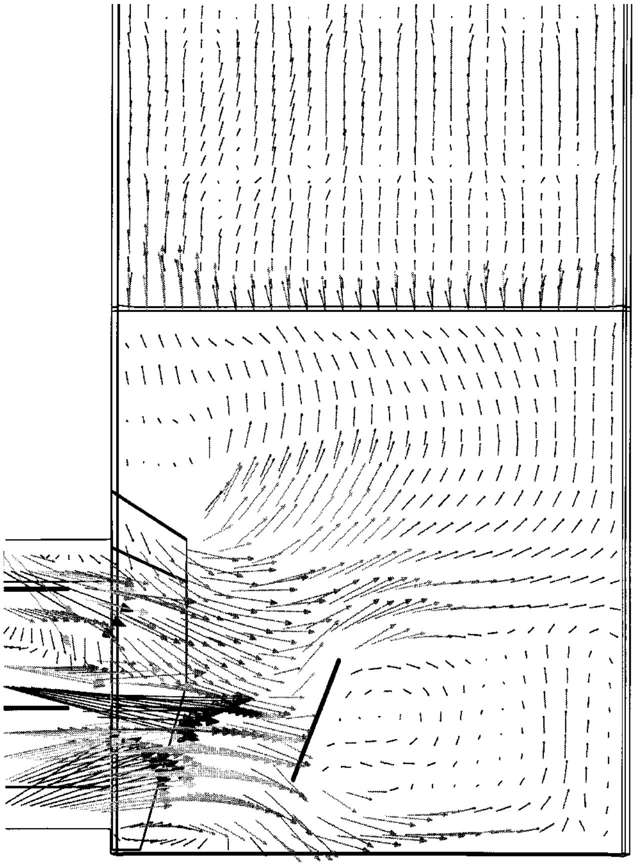

Figure 11 Vector plot; final design.

Results are presented in Figure 12 for a four down-comer arrangement with two feedwater pipes and in Figure 13 for a three downcomer arrangement with one feedwater pipe. The figures are vector plots of a horizontal section and display the flow distribution, basic geometry, and outlines of the cyclone separators. The plots focus on regions near the downcomer which is illustrated by the converging flow pattern.

Figure 14 is a plot of thermal stress for a section of the shell and drum head which was used for a life evaluation study. The plot depicts Von Mises stresses and ranges from 4,000 psi to 22,000 psi.

The numerical results in conjunction with other analyses provided an optimized downcomer and feedwater pipe arrangement. The result was a substantial cost savings while maintaining all functional aspects of the design. This example illustrates how numerical modeling can be used to augment traditional analysis techniques.

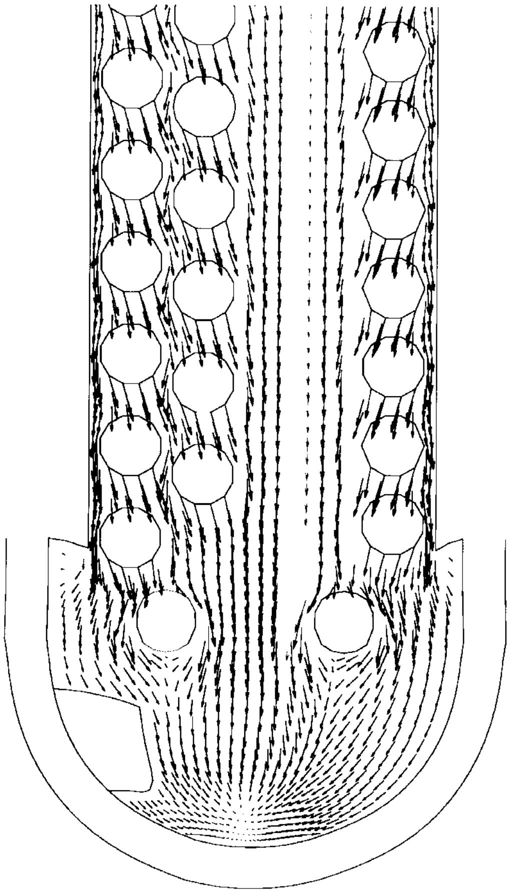

Figure 12 Vector plot; four downcomer arrangement.

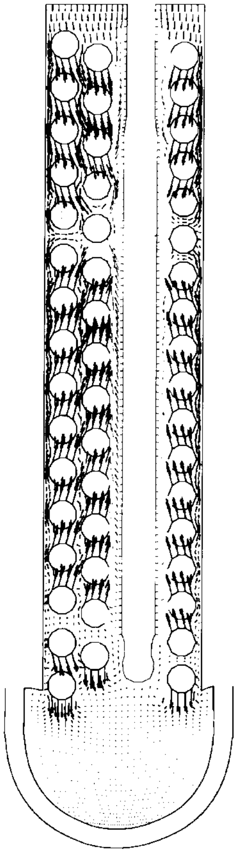

Figure 13 Vector plot; three downcomer arrangement.

Present Limitations & Future Capabilities

A numerical analysis begins by subdividing the problem geometry into a finite number of control volumes. For three-dimensional problems, hexagonally shaped elements are arranged for this purpose.[7, 8] Present technology limits 3-D grid generation by constricting the grid, either entirely or in parts, to ordered indices (I, J, K array indices are convenient for digital computer matrix solutions). For complicated geometries, this limitation can make grid generation difficult. Future models will no longer have this limitation as so-called unstructured (without explicit I, J, K order indices) grid generators and solvers are an active area of development for commercially available and in-house proprietary codes.

Figure 14 3 downcomer – feedwater to center down-comer Von Mises stress (psi).

One problem which plagues today's numerical predictions is estimating the job schedule. The methodology with most numerical analyses is to attempt alternate arrangements until an acceptable flow solution is obtained. This results in vastly different time tables for completing each analysis. One analysis may require three or four alternates before finding an acceptable arrangement while a similar analysis may require twenty. Experience largely determines the direction taken from one alternate arrangement to the next. However, most problems are unique in some regard which complicates the issue.

The future of numerical modeling is a broad subject and ties computer software to hardware. Today's models are limited by both and as hardware becomes faster and more economical, software developers follow with additional capabilities. In addition, added complexity improves the accuracy of the predictions at the expense of execution time. Since the purpose of this paper is to discuss applications, additional background information to solution techniques is not discussed.

Conclusions

Several successes have been demonstrated for isothermal non-reacting flow problems. Examples of these are the pulverizer windbox and scrubber applications discussed previously. Combustion solutions have yielded reliable predictions for flow, temperature, heat flux,[9] and major gas species but can be limited by the prediction of pollutant species. More complicated flow problems such as those involving two fluid models (nozzles discharging into atmosphere) or complicated chemistry are limited to specialized programs developed for a specific problem.

Due to the cost effectiveness and successes of the past, increased software capability, and more economical computers, numerical modeling will continue to grow in the power industry. If history is an indication of the future, one could easily conclude that tomorrow's numerical applications will be completed on virtually every major boiler component.

References

1. White, F.M., Viscous Fluid Flow, 2nd ed., McGraw-Hill, New York, 1991.
2. Latham, C.E., et al., "Designing Air Staging Systems With Mathematical Modeling," Presented at the AFRC/JFRC Pacific Rim International Conference on Environmental Control of Combustion Processes, Maui, Hawaii, October 16-20, 1994.
3. LaRose, J.A., and M.W. Hopkins, "Numerical Flow Modeling of Power Plant Windboxes," Presented at the Power-Gen Americas '95, Anaheim, California, December 5-7, 1995.
4. Meyers, R.A., ed., Handbook of Synfuels Technology, McGraw-Hill, New York, 1984.
5. Stultz, S.C., and J.B. Kitto, eds., Steam/its generation and use, 40th ed., The Babcock & Wilcox Company, Barberton, Ohio, 1992.
6. JAC Humphrey, "Solid Particle Erosion in Turbulent Flows Past Tube Banks," Oak Ridge National Laboratory, February 1990.
7. Advanced Scientific Computing, TASCflow User Documentation, Waterloo, Ontario, Canada, January 1993.
8. PDA Engineering, P3/PATRAN User Manual, Costa Mesa, California, December 1993.
9. Kitto, J.B., and M.J. Albrecht, "Elements of Two-Phase Flow in Fossil Boilers," Two-Phase Flow Heat Exchangers, S. Kakac, et al. eds., Kluwer Academic Publishers, Norwell, Massachusetts.