A COMPARISON OF COMPUTATIONAL AND EXPERIMENTAL METHODS FOR DETERMINING THE GAS FLOW PATTERNS IN THE KRAFT RECOVERY BOILER

A. K. JONES, T. M. GRACE, AND T. E. MONACELLI

JUNE, 1988
A Comparison of Computational and Experimental Methods for Determining the Gas Flow Patterns in the Kraft Recovery Boiler

A. K. Jones, T. H. Grace, and J. K. Monacelli

Portions of this work were used by AKJ as partial fulfillment of the requirements for the Ph.D. degree at The Institute of Paper Chemistry. This is to be presented at the TAPPI Engineering Conference in Chicago on September 19-22, 1988

Copyright, 1988, by The Institute of Paper Chemistry

For Members Only

NOTICE & DISCLAIMER

The Institute of Paper Chemistry (IPC) has provided a high standard of professional service and has exerted its best efforts within the time and funds available for this project. The information and conclusions are advisory and are intended only for the internal use by any company who may receive this report. Each company must decide for itself the best approach to solving any problems it may have and how, or whether, this reported information should be considered in its approach.

IPC does not recommend particular products, procedures, materials, or services. These are included only in the interest of completeness within a laboratory context and budgetary constraint. Actual products, procedures, materials, and services used may differ and are peculiar to the operations of each company.

In no event shall IPC or its employees and agents have any obligation or liability for damages, including, but not limited to, consequential damages, arising out of or in connection with any company's use of, or inability to use, the reported information. IPC provides no warranty or guaranty of result.
A DESCRIPTION OF FLUIDS

A. E. Jones and T. M. Grace
The Institute of Paper Chemistry
Appleton, WI 54912

J. E. Nasselle
Babcock & Wilcox
Power Generation Group
Barberton, Ohio 44203

ABSTRACT

Computational fluid dynamics was applied to the problem of gas flow in a draft recovery furnace. For simplicity, the case chosen to be modeled is the "cold flow case" (the gas flow pattern that arises directly from the method of air introduction and furnace geometry, without complications from combustion, temperature distributions, and liquor spray interactions). The framework for the computational model was FLUENT (a commercially available, finite-difference, computational fluid-flow program). A three-dimensional description of the recovery furnace with 50,000 computational cells was used. Neutral symmetry of the furnace was assumed to reduce the computational effort by a factor of two. The results of the computational model are compared with experimental cold flow data obtained on a 1/8th scale model of a recovery furnace. The analytical model corresponded closely with the experimental cold flow data.

INTRODUCTION

This paper describes the first stage in the development of a computational model for the recovery furnace, namely, the simulation of cold flow in an existing furnace design. In a cold flow model only the geometry and method of introduction of the air are assumed to be important. Neglected are the effects of temperature, combustion, and in this case the interaction between the black liquor spray and the gas phase. A particular 286 scale furnace located in Debilder, IA, was simulated, as data were available for comparison [1]. 904 measured the velocity profiles in a 1/8th scale replica of this furnace design, under a number of flow conditions. This paper examines two computationally developed cold flow patterns in this furnace obtained by using FLUENT (a commercially available computational fluid-flow program), and compares them with two experimentally determined flow patterns.

The computational flow field is found by use of an numerical cold flow model of the Debilder unit at 273K. The experimental flow field was developed in a 1/8th scale model of this same recovery furnace. The NME report then provides scaling constants for converting the velocity data to the full sized furnace at 417K. These velocities are then corrected to 273K in order to make a comparison to the computational work.

A DESCRIPTION OF FLUIDS

The main tool used in the computational simulation was FLUENT, a finite difference program for the modeling of fluid flows. FLUENT is flexible and comprehensive, permitting its use in a wide variety of flow situations, including two or three dimensional flows, laminar or turbulent flows, and swirling flows. Additional features of FLUENT include a air-flow radiation model, the PSI-CELL model for two-phase flow (2), a combustion model, a number of turbulence models, and a porous flow model.

The variables that are solved for in the simplest three-dimensional case are velocity, - velocity, v-velocity, and pressure. If temperature effects are added, the enthalpy and the radiative heat fluxes are also determined. If the combusti- lence model is used, then the kinetic energy of turbulence and the dissipation rate are found. If combustion is included, the mole fractions of combustibles (Q), oxygen (O), and products (P), must be calculated in each cell. The maximum number of finite element cells that can be solved for in the version of FLUENT used (version 2.8) is 50,000.

The attractive features of FLUENT are 1. Interactive input. 2. Flexibility - almost any type of flow can be modeled. 3. Internally generated relaxation coefficients, which usually result in stable iterations; and 4. Excellent graphics for viewing the results.

The process of setting up a fluid flow simulation is straightforward, as the description of a flow is menu driven. Cartesian or cylindrical regions can be described. In a cartesian description the cells are rectangular parallelepipeds that can be defined by coordinate directions. A number of different cell types can be specified. Typically, three types of cells are used in this model:

1. live cells - the variables in these cells are solved for at each iteration; 2. inlet cells - the values of the flow variables are specified for these cells and remain the same throughout the calculation; 3. wall cells - these cells act as barriers to the flow and are used to describe the flow geometry; 4. outlet cells - the value of the flow variables are solved for, but these cells are placed hundreds of miles from the flow. 5. symmetry cells - they define lines or planes of symmetry in the flow, which reduces the number of cells necessary to describe a given flow geometry.

In addition to a description of the flow geometry, it is necessary to specify the physical properties of the fluid and the boundary conditions in the wall and inlet cells. The mass flow rates in the inlet cells are controlled by varying the gas velocity or the distance between adjacent nodes.

THE COLD FLOW SIMULATION

The analytical "cold flow" model was set up with physical dimensions as close as possible to those of the full-sized furnace, in order to justify comparison between the experimental and analytical cases. The upper limit of 50,000 nodes makes it
necessary to use some approximations. An isometric view of the furnace is shown in Fig. 1 and 2. Due to the symmetry involved, it is only necessary to describe half of the furnace. The flow patterns in the other half of the furnace will be a mirror image of the area of the furnace modeled. The symmetry plane is parallel to the side walls and cuts the bullnose and bed in half.

Fig. 1 Illustration of furnace geometry low velocity territories.

Fig. 2 Illustration of furnace geometry high velocity territories.

The full-sized furnace is 9.6 meters by 10.6 meters in cross section and 25.0 meters high (to the bullnose). The PLANT model is 10.0 meters by 5.0 meters (due to symmetry) and 30.0 meters high. The bullnose extends 5 meters across the top of the furnace in both cases. The bullnose slopes slightly downward to the back of the furnace, but this was ignored in the computational model in order to reduce the number of nodes used. The resulting error on the gas flow patterns should be negligible, as the slope is minimal.

The air inlets were modeled in the following manner:

**Primary Air**

A row of nodes along the walls were designated as inlets; the distance between the inlet nodes and the nodes above and below were adjusted in order to satisfy the volumetric flow rates. This treats the primary air as if it was a plug jet completely enroiling the furnace. A shortage of nodes made description of individual primary ports impractical. The primary air was injected 1 meter above the furnace floor at a velocity of 40 meters/sec.

**Secondary Air**

Individual air ports of approximately the same dimensions as the ports in the actual furnace were used for the secondary air. The dimensions of the ports in the full-sized furnace were 5.6 cm by 55.5 cm. In the model they were 11.6 cm by 33.7 cm. The areas were the same; the individual diameters had to be adjusted. This approximation was necessary due to computer hardware limitations that restricted the total number of cells that could be used. The width of the secondary and tertiary air ports (11.6 cm) was an average of the actual widths. The half of the furnace modeled had 14 secondary ports, each with an inlet velocity of 36 meters/sec (identical to the prototype furnace). The injection level was 3 meters above the furnace floor.

**Tertiary Air**

Separate air ports on the walls were used, with approximately the same shape as those in the actual furnace. Two configurations were examined. In the first configuration 4 ports at 10 meters above the furnace floor were used, two on each the front and back walls, with an inlet velocity of 10 meters/sec (identical to the prototype furnace), as shown in Fig. 3. The ports were 11.4 cm by 1.02 m, the same area as the actual ports which were 15.7 cm by 76.2 cm. In the second configuration, 8 smaller ports with an inlet velocity of 57 meters/sec were used, four on each the front and back walls. Two closely spaced levels of tertiary air were used (at 9.5 and 6.5 meters above the floor of the furnace), this is shown in Fig. 4. The port sizes in this case were 11.4 cm square, as opposed to the 12.7 cm 10 circular ports used in the actual furnace.

**Fig. 3 Low velocity territories.**

**Fig. 4 High velocity territories.**

Two computational cases were examined: the first corresponds to experimental case 1, with the low velocity territories; the second to experimental case 2, with the high velocity territories. The computational cases were set up so that they corresponded as closely as possible to the two experimental cases.
The bed shape used (Fig. 5) corresponded as closely as possible to the one used in the experimental work (Fig. 6); with the top of the bed extended slightly above the secondary air ports. The use of body fitted coordinates would eliminate the staggered bed shape and the resulting numerical problems, leading to a more accurate description of the flow around the bed, but this is not yet available as part of PLUNIT.

Fig. 5 Outline of computational bed model.

The mass flow rates in the PLUNIT models were the same as the full-sized BWR furnace; the total flow rate for the half of the furnace modeled was 60 kg/sec (380,000 lb/hr). In case 1 the flow was split into 50% primary, 40% secondary and 10% tertiary air. In case 2 the flow was split into 40% primary, 40% secondary and 16% tertiary air.

EXPERIMENTAL DATA

The experimental data used for comparison with the analytical cold flow models were part of a NUREG investigation (1). The furnace used was the Hanford TRIGA Reactor Nurtury Pluton, located in Richland, WA. This furnace was experiencing excessive particulate carryover. The problem was believed to be due to insufficient breakup of a high velocity core of gas by the existing tertiary air jets.

A number of changes in the tertiary air system were considered. These changes were evaluated by constructing a 1/8th scale replica of the PINDRED recovery furnace, and conducting cold flow measurements in this replica. The velocities within the scaled-down furnace were measured with a hot film anemometer at two traverse planes. T1 located halfway between the secondary and tertiary air ports, and T2 located about halfway between the tertiary air ports and the bulbous. Low-velocity plots were constructed using these data.

The two cases that were simulated will be designated case 1 and case 2. In both cases a bed was included in the bottom of the furnace; the shape of this bed is shown in Fig. 6.

RESULTS - A COMPARISON BETWEEN THE EXPERIMENTAL AND COMPUTATIONAL CASES

This section will describe in detail the results of modeling with PLUNIT; the two cases described earlier and compare these results with the experimental cases. The contour plots will be compared on the basis of average upward velocity at 273K, and on the general flow pattern.

In order to make a comparison of the experimental and computational cases it is first necessary to put them on an equal basis. In the experimental work the average upward velocity is calculated by adding together all the positive velocities and dividing by the number of these positive velocities. The average upward velocity in the prototype (full-size furnace at 60°F) can then be calculated based on scaling criteria. In case 1 the scaling factor is 2.5, in case 2 it is 2.5. This results in an average velocity in feet/min which can then be converted to m/sec. Finally, a temperature correction is applied converting the experimental results to 273K. The results obtained computationally are already in m/sec, at the cold flow temperature of 273K.

Case 1 - CI Traverse

The CI traverse is a horizontal slice located above the bed as shown in Fig. 1. Contours of the upward velocity (V-velocity) are shown in Fig. 7 and 8 (experimental and computational results).

Fig. 7 Low velocity territies - CI slice experimental results.

The main features of these contour plots are the upflow region in the center of the furnace and the downflow regions around the outside. The computational CI traverse for case 1 shows good agreement with the experimental results with respect to the general flow patterns, with the location of downflow regions and the maximum upward velocity corresponding to the experimental results. The average upward velocity is about 40% higher in the experimental case than in the computational case (2.40 m/sec vs. 1.68 m/sec). This is due to the use of a staggered bed; the air jets tend to be
deflected upward rather than following the surface of the bed. This reduces the intensity of the central core, as some of the gas in deflected upward before it reaches the center of the furnace.

The velocities are highest in the center as all the secondary jets meet at this point and are forced to move upward. The downdraft regions around the perimeter of the furnace are created due to entrainment of gas by the secondary jets.

**Case 1 - T2 Traverse**

At the T2 traverse, a horizontal slice located above the tertiary jets (see Fig. 1), the velocity contours obtained experimentally here have taken on a much different appearance. This is shown in Fig. 9. At the back of the furnace a region of recirculation is present. As the front of the furnace the upward velocities are large. The back of the furnace is a large stagnant region, resulting in poor use of the furnace volume for combustion of the fine gases.

**Fig. 9 Low velocity tertaries - T2 slice experimental results.**

At the T2 traverse in case 1 the existence of a large stagnant region seen experimentally is confirmed using PIV/PIVF (Fig. 10). The location of the maximum upward velocity is at the front wall in both cases. The average upward velocity in the experimental case is 1.50 m/sec, compared to 0.83 m/sec in the computational case. This is once again a result of the numerical problems discussed earlier, persisting up to the T2 traverse.

**Fig. 10 Low velocity tertaries - T2 slice.**

**Case 2 - T1 Traverse**

The comparison at the T1 traverse for case 2 is essentially the same as for case 1, except that the discrepancy in the average upward velocity is higher (1.60 vs. 2.94 m/sec) (Figs. 11 and 12).

**Fig. 11 High velocity tertaries - T1 slice experimental results.**

At the T1 traverse in case 2 has much higher velocity tertaries; this results in a better utilization of the furnace volume, as shown in Fig. 13. The stagnant region at the back of the furnace has been eliminated.

**Fig. 12 High velocity tertaries - T1 slice.**

**Case 2 - T2 Traverse**

Case 2 has much higher velocity tertaries; this results in a better utilization of the furnace volume, as shown in Fig. 13. The stagnant region at the back of the furnace has been eliminated.

**Fig. 13 High velocity tertaries - T2 slice experimental results.**

The experimental T2 traverse in case 2 has no downdrafts; the computational results have only small areas of downdraft against the side walls (Fig. 14). The average upward velocity found experimentally and computationally are both 0.80 m/sec. The numerical problems are no longer a factor as the center core has been almost completely eliminated in both cases, and the average upward velocity should be just the total mass flow rate divided by the cross sectional area times the density; which is the same in both cases.

The reason for the elimination of the stagnant region can be seen by comparing Fig. 15 and 16. Contours of the U-velocities are shown on a horizontal slice that cuts through the midpoint of the tertiary jets. Immediately apparent is the difference in penetration distance between the two
The main limitations of computational model-

1. the need to use staggered grids to describe a diagonal surface (i.e., bed surface);
2. the requirement of built-in turbulence models;
3. the large amount of computer time required to converge complex problems; (a typical problem takes about 7 days of CPU time to converge on a MicroVAX); b. the difficulty in deciding when a flow pattern has converged; and c. the limitation in the number of nodes, making it impossible to describe fine details of the flow around the bed and in jets.

Experimental scale modeling of gas flows has the following limitations: 1. once a scale model has been constructed, only what change can be made in the geometry; 2. it is difficult or impossible to satisfy all the scaling criteria; 3. it is very difficult to measure large numbers of velocities within the model, and these velocities may be altered by the act of measuring them; a temperature effect cannot be easily added.

CONCLUSIONS

The main conclusions that can be drawn from the modeling effort thus far are

1. FLUENT can be used effectively to simulate the flow patterns in the kraft recovery boiler;
2. the benefit of modifying the tertiary jets in the new furnace can be observed by the use of FLUENT, in agreement with experimental results;
3. numerical problems were created due to the primary and secondary air jets impacting on the staggered bed. In future modeling work the bed will not extend above the level of the secondaries. This should result in a more realistic flow pattern.

ACKNOWLEDGMENT

Portions of this work were used by AEI as partial fulfillment of the requirements for the M.D. degree at the Institute of Paper Chemistry.

LITERATURE CITED