HEAT TRANSFER BY PARTICLE CONVECTION
AT THE WALL OF A CIRCULATING FLUIDIZED-BED

by

Peter Daniel Noymer

M.S., Mechanical Engineering, Massachusetts Institute of Technology (1992)
B.S.E., Mechanical and Aerospace Engineering, Princeton University (1988)

Submitted to the Department of Mechanical Engineering
in partial fulfillment of the requirements for the degree of

DOCTOR OF PHILOSOPHY IN MECHANICAL ENGINEERING

at the Massachusetts Institute of Technology, June 1997.

© 1997, Massachusetts Institute of Technology. All rights reserved.

Signature of Author

Department of Mechanical Engineering
June 1997

Certified by

Leon R. Glicksman
Professor of Architecture and Mechanical Engineering
Thesis Supervisor

Accepted by

Ain A. Sonin
Chairman, Graduate Committee
Department of Mechanical Engineering

JUL 21 1997
HEAT TRANSFER BY PARTICLE CONVECTION
AT THE WALL OF A CIRCULATING FLUIDIZED-BED

by

Peter D. Noymer

Submitted to the Department of Mechanical Engineering on May 2, 1997,
in partial fulfillment of the requirements for
the degree of Doctor of Philosophy in Mechanical Engineering.

ABSTRACT
Fluidized-bed combustors are gaining popularity as environmentally-sound options for burning
otherwise "dirty" fuels, such as coal. A fluidized-bed combustor can be used as the boiler for
steam production in a power-generation system, and a circulating fluidized-bed (CFB) is one type
of fluidized bed used for combustion. When modeling heat transfer at the peripheral walls of a
CFB, it becomes apparent that the hydrodynamic characteristics of the gas-solid flow are highly
influential. In this thesis, a number of hydrodynamic characteristics have been observed and
measured, and their effect on heat transfer has been analyzed. Furthermore, analytical and
computational modeling of the observed phenomena has provided a better understanding of near-
wall hydrodynamics.

Utilizing a novel flow-visualization technique, measurements have been made of the residence time
and velocity of the bed material at the wall of the CFB. These measurements have been made in a
scale-model CFB, which can be shown to be hydrodynamically similar to a full-sized CFB.
Average cluster velocities were measured in the range of 1.1-1.2 m/s, with little dependence seen
on any of the operating conditions. Average cluster-wall contact times were measured in the range
of 0.15-0.50 s, with a strong dependence on solid density and particle size observed. The relative
solid-to-gas density ratio is one important parameter that differs between atmospheric and
pressurized CFBs. The clusters apparently accelerate to a terminal velocity as they descend along
the wall, and the acceleration can be shown to be an important consideration for particle-convective
heat transfer. Also, the cluster-wall contact times are distributed according to a gamma probability
density function for a given set of operating conditions. This information can be used to analyze
and predict the thermal development region for particle-convective heat transfer.
Some basic modeling methods were employed in order to understand the observations of cluster velocities and cluster-wall contact times. A review of prior literature indicates that the lift and drag forces on a permeable body near a wall may be significantly different than those on a solid body in unbounded flow, and these forces may govern the observed phenomena. Modeling a cluster as a circular cylinder, steady flow past permeable cylinders has been investigated using a commercially-available computational fluid dynamics program. Three dimensionless parameters are considered: the Reynolds number based on the diameter of the cylinder, the ratio of the permeability of the cylinder to the square of the cylinder diameter, and the ratio of the cylinder-wall gap to cylinder diameter. Typical values for these parameters in a CFB are a Reynolds number of about 1000, a gap ratio of about 0.01 and a permeability ratio of about $10^{-4}$ (corresponding to a solid fraction of about 8%). Under these conditions, for example, a 100-200% increase in drag and a lift coefficient of about 0.5 can be computed. These results are in general agreement with the limited amount of existing experimental and computational data for similar configurations. The results for drag explain the observations of slower cluster velocities in CFBs, and the results for lift agree with the magnitude and parametric variations observed for cluster-wall contact times.

Thesis Supervisor: Leon R. Glicksman
Title: Professor of Architecture and Mechanical Engineering
ACKNOWLEDGEMENTS

This work was sponsored by the National Science Foundation, Chemical and Transport Systems Division, under grant #CTS 93-03695. I am grateful to MIT/Lincoln Laboratory, the Fluid Mechanics Lab and the Welding Lab at MIT for the use of their equipment; to Professor Qingyan Chen and Weiran Xu for their guidance in performing computational studies; to Professor Kenneth Breuer and Ruben Rathnasingham for their assistance in wind-tunnel testing; and to Anand Devendran, Sushil Panta and Jeremy Godinger for providing assistance in this project.

I would like to thank Professor Leon Glicksman, my advisor, for his steadfast support, technical advice, and professional guidance throughout my doctoral program. His constant input and enthusiasm for research kept me focused on the successful completion of this project. I would also like to thank the members of my thesis committee: Professors Bora Mikic, Peter Griffith and Janos Beer. Their expertise, oversight and positive reinforcement were invaluable. Thanks also go out to Professor John Lienhard for rounding out my academic experience and providing me the opportunity to develop my teaching skills.

One of the positive aspects of being at MIT is the opportunity to work and interact with some incredibly intelligent and interesting people. I consider my closest colleagues to be in this category: Paul Farrell, Stefano Schiaffino, Matt Hyre, Richard H. Tan and Steven Treppo. I owe special thanks to Paul for enhancing my education both in the laboratory and in the classroom, and for being my lunch date on many occasions. I also owe special thanks to Stefano for setting an unbeatable standard for research motivation and for his unique personality.

I will always be to grateful to the members of my family for their unwavering love and support. My parents, Barbara and Arthur Noymer, have stood behind me and provided encouragement throughout my life. My siblings have had some impact on me – less than they'd like to think, but I'll admit that it's probably more than I'd like to think. Finally, Grace Colón – now my fiancée and soon to be my wife – deserves a lifetime of thanks for providing constant comfort and inspiration through both the ups and downs of my studies.
# TABLE OF CONTENTS

**TITLE** .......................................................................................................................... 1  
**ABSTRACT** ......................................................................................................................... 3  
**ACKNOWLEDGEMENTS** ...................................................................................................... 5  
**TABLE OF CONTENTS** ...................................................................................................... 6  
**NOMENCLATURE** .................................................................................................................. 9  

1. **INTRODUCTION: FLUIDIZATION AND FLUIDIZED-BED BOILERS**  
   1.1 A GENERAL DESCRIPTION OF FLUIDIZATION ......................................................... 11  
   1.2 REGIMES OF FLUIDIZATION ..................................................................................... 12  
   1.3 APPLICATIONS FOR FLUIDIZED BEDS ..................................................................... 15  
   1.4 FLUIDIZED-BED BOILERS ......................................................................................... 16  
   1.5 REFERENCES ............................................................................................................. 22  

2. **HYDRODYNAMICS AND HEAT TRANSFER IN CFBS**  
   2.1 A DESCRIPTION OF CFB HYDRODYNAMICS ......................................................... 23  
   2.2 HEAT TRANSFER IN CFBS ......................................................................................... 28  
   2.3 SCALE MODELING OF CFBS .................................................................................... 36  
   2.4 THESIS PROJECT GOALS ......................................................................................... 40  
   2.5 REFERENCES ............................................................................................................. 41  

3. **EXPERIMENTAL APPARATUS AND DATA REDUCTION**  
   3.1 OVERVIEW .................................................................................................................. 45  
   3.2 SPECIFICATIONS FOR THE SCALE-MODEL CFB .................................................... 46  
   3.3 DESCRIPTION OF EXPERIMENTAL METHODS ....................................................... 52  
   3.4 DATA ACQUISITION AND REDUCTION .................................................................... 58  
   3.5 REFERENCES ............................................................................................................. 67  

4. **EXPERIMENTS: RESULTS AND DISCUSSION**  
   4.1 EXPERIMENTAL OPERATING CONDITIONS ......................................................... 69  
   4.2 MEASUREMENTS OF CLUSTER VELOCITIES ......................................................... 70  
   4.3 MEASUREMENTS OF CLUSTER-WALL CONTACT TIMES ....................................... 74  
   4.4 ANALYSIS OF CLUSTER MOTION AND PARTICLE CONVECTION ....................... 79  
   4.5 SUMMARY OF EXPERIMENTAL RESULTS AND ANALYSIS ............................... 86  
   4.6 REFERENCES ............................................................................................................. 87
5. MODELING: RESULTS AND DISCUSSION

5.1 OVERVIEW ........................................................................................................... 89
5.2 ANALYTICAL MODELING ................................................................................... 92
5.3 COMPUTATIONAL MODELING ............................................................................ 107
5.4 SUMMARY OF MODELING RESULTS ................................................................ 128
5.5 REFERENCES ....................................................................................................... 129

6. CONCLUSIONS AND RECOMMENDATIONS

6.1 SUMMARY OF RESULTS AND CONCLUSIONS .............................................. 133
6.2 RECOMMENDATIONS FOR FUTURE WORK .................................................... 137

A. Summary of particle characteristics ................................................................. 139

B. Design data for pressure transducers ............................................................... 141

C. Probability analysis for data reduction ......................................................... 143

D. Calibration of the TIV system for temperature measurements .................. 145

E. Input files for computational simulations .................................................... 153
NOMENCLATURE

\( A \) \quad \text{area} \\
\( c \) \quad \text{specific heat capacity, volumetric solid concentration (App. D only)} \\
\( d_p \) \quad \text{particle diameter} \\
\( D \) \quad \text{diameter} \\
\( f \) \quad \text{fraction of wall covered by clusters} \\
\( g \) \quad \text{gravitational constant} \\
\( G_s \) \quad \text{solid recycle flux} \\
\( h \) \quad \text{heat transfer coefficient} \\
\( k \) \quad \text{thermal conductivity} \\
\( L \) \quad \text{bed height} \\
\( m \) \quad \text{mass} \\
\( n \) \quad \text{parameter in gamma probability function} \\
\( p \) \quad \text{probability density function} \\
\( P \) \quad \text{pressure} \\
\( q \) \quad \text{heat flux} \\
\( R \) \quad \text{radius} \\
\( Re \) \quad \text{Reynolds number} \\
\( t \) \quad \text{time} \\
\( T \) \quad \text{temperature} \\
\( u \) \quad \text{velocity} \\
\( v \) \quad \text{velocity} \\
\( x \) \quad \text{axial distance} \\
\( y \) \quad \text{lateral distance} \\
\( \alpha \) \quad \text{thermal diffusivity} \\
\( \varepsilon \) \quad \text{volumetric void fraction, emissivity (App. D only)} \\
\( \eta \) \quad \text{dimensionless time} \\
\( \zeta \) \quad \text{dimensionless distance} \\
\( \mu \) \quad \text{dynamic viscosity} \\
\( \nu \) \quad \text{kinematic viscosity} \\
\( \rho \) \quad \text{density} \\
\( \kappa \) \quad \text{permeability, extinction coefficient (App. D only)} \\
\( \tau \) \quad \text{time, transmissivity (App. D only)}
1. INTRODUCTION: FLUIDIZATION AND FLUIDIZED-BED BOILERS

"Fluidization is the operation by which fine solids are transformed into a fluid-like state through contact with a gas or liquid ... fluidization engineering is concerned with efforts to take advantage of this behavior and put it to good use."

[Kunii & Levenspiel, 1969]

1.1 A GENERAL DESCRIPTION OF FLUIDIZATION
The process of fluidization involves passing a fluid – a liquid or a gas – through a quantity of solid particles contained in some chamber. The chamber containing the particles is often called a "riser", and the quantity of particles is commonly referred to as a "bed" of particles, as shown in Figure 1.1 below. If the fluid velocity is high enough, the flow will segregate the particles somewhat and the bed will begin to qualitatively behave like a fluid. This gives rise to the term "fluidized bed" for a chamber filled with particles through which a fluid is passed at or above a certain velocity. In general, the reason for doing this is to allow contact between the fluid and the solid material in the interest of promoting an interaction between the two. Initial commercial development of fluidized beds focused on chemical reactions, in which the solid material was a catalyst for some desired chemical transformation of the fluid. In the interest of brevity, the remainder of the discussion on fluidization will focus on gas-solid systems only, since the research reported herein relates only to gas-solid fluidized beds.

![Diagram of fluidization](image)

**FIGURE 1.1:** The general concept of fluidization
1.2 REGIMES OF FLUIDIZATION

The fluid-like behavior of the bed of solid particles can vary greatly, depending on the gas superficial velocity (volume flow rate per unit area), and in some cases on the flow rate of the solid particles themselves. The observed behavior can be divided into distinct "regimes" of fluidization, in which each regime corresponds to a different range of superficial velocities. A picture showing the qualitative differences between the major regimes is given in Figure 1.2.

![Regimes of fluidization diagram]

**FIGURE 1.2: Regimes of fluidization**

1.2.1 Packed beds

In the lower limit of superficial velocities, the gas merely trickles through the bed of particles, giving rise to the term "packed bed". The flow rate of the gas is so low that the particles remain relatively undisturbed and the gas simply flows through the interstitial gaps between the particles. A packed bed is not fluidized, strictly speaking, because it is the segregation of particles that creates the fluid-like behavior. At this extreme end of the spectrum of superficial velocities, one can imagine that the packed bed functions much like a filter.

1.2.2 Minimum fluidization

As the superficial velocity is increased somewhat, the fluid has sufficient momentum to move the particles, and the drag between the fluid and the solid particles stirs things up a bit. When the superficial velocity is high enough, there is just enough gas momentum to overcome the viscous losses from suspending the bed of particles. This superficial velocity is known as "incipient" or "minimum" fluidization. In fact, the minimum fluidization velocity is defined as the superficial velocity at which the pressure drop of the gas is equal to the weight of the bed of particles. The minimum fluidization velocity can be measured, and it is a characteristic of solid particles of a
certain density and size and a particular gas in a given thermodynamic state (i.e., with a given temperature and pressure or with a given density and viscosity).

1.2.3 Bubbling beds
Increasing the superficial velocity somewhat beyond that for minimum fluidization, one observes the extra gas flow in the form of void pockets or "bubbles". This regime of fluidization is referred to as the bubbling regime, and the bed of particles visibly takes on fluid-like behavior. A bubbling fluidized-bed (BFB) looks very much like a pile of boiling mud – basically one sees bubbles rising through what appears to be a very heavy fluid, but is in fact a pile of granular particles. The onset of bubbling in a fluidized bed typically occurs at a superficial velocity slightly higher than that for minimum fluidization; with some materials, the two can be coincident [Grace, 1982].

1.2.4 Fast fluidization
At much higher multiples of the minimum fluidization velocity, perhaps 100 times or more, the regime called fast fluidization is encountered. The momentum flux of the gas is so high in this case that most of the particles in the bed will be entrained and dragged out of the riser. In order to maintain a steady state in this regime, particles must be introduced into the bottom of the riser at the same rate at which they are being dragged out. Interestingly enough, such a condition causes the definable "bed" of particles to disappear into a much more disperse concentration of particles.

Fast fluidized beds can also be categorized by the rate at which solid particles are introduced and whether or not those solid particles are coming from a fresh source or are being recycled from the riser discharge. A pneumatic transport line, for example, is characterized by a relatively high superficial velocity and a relatively low flux of solid particles, and the solid particles being supplied are fresh and meant to be transported elsewhere. A circulating fluidized bed (CFB), on the other hand, will have a recirculation loop to return the particles from the discharge of the riser to the entrance of the riser. CFBs typically run at lower superficial velocities or higher rates of solid replenishment than pneumatic transport lines. Figure 1.3 shows a schematic for a typical CFB.

1.2.5 Summary of fluidization regimes
As mentioned earlier, the primary motive for fluidization is to promote an interaction between the solid particles and the fluid. Each of the flow regimes mentioned above has relative advantages and disadvantages, depending on the application. In a packed bed, for example, the solid particles are not moving and the contact area of the fluid with the solid is limited by the fact that the solid particles are also in contact with each other. Elevating the superficial velocity toward and above minimum fluidization alleviates that, and into the bubbling regime and beyond one can take
advantage of the vigorous mixing either from the motion of the bubbles or the vigorous mixing caused by the gas flow. However, at higher superficial velocities more power is required for air delivery and the systems may become more complex and therefore potentially more difficult to design, analyze and even control.

![Diagram of a circulating fluidized-bed system]

**FIGURE 1.3**: Schematic for a typical circulating fluidized-bed
1.3 APPLICATIONS FOR FLUIDIZED BEDS
In his book on gas-solid fluidization, Geldart [1986] divides the applications of fluidized beds into two broad categories: those promoting physical interactions, such as heat and/or mass transfer; and those promoting chemical reactions, in which the solid may be the catalyst or the primary reactant. The different types of fluidized beds described earlier find use in both types of applications, and some of the more common ones are discussed next.

1.3.1 Packed beds
Although packed beds are not fluidized, they present an interesting point from which to begin the discussion of applications of gas-solid reaction systems. Packed beds typically find utilization in catalytic and energy storage applications. The porous matrix that one finds in an automotive catalytic converter is exactly analogous to a packed bed. In this application, exhaust gas from an automobile engine is passed through a device which has narrow passages. These passages provide a relatively high surface-to-volume ratio for the catalytic solid material to react with the flowing exhaust gas. Similarly, one might find a porous matrix in a Stirling engine, for example. In this application, heat is alternatively stored and removed as the working fluid is moved by a piston. The porous matrix also offers a high surface-to-volume ratio to absorb or reject the heat, as well as a relatively high heat capacity such that little heat is lost between stages of a cycle.

1.3.2 Petroleum and chemical industries
Early fluidized beds were designed as chemical reactors, primarily for the petroleum industry in the production of gasoline and the regeneration of catalysts required for refining. Fluidized beds have found subsequent use in many other chemical processes, including the production of polymers and the application of coatings to manufactured goods.

1.3.3 Energy and power generation
More recently, the physical and chemical benefits of fluidized beds have prompted their use in energy and power generation systems. Coal is a common fuel used in boilers, and the steam produced from these boilers can be used in industrial processes or for generating electrical power. Harmful gases can be emitted during the combustion of coal, including oxides of nitrogen (NOx) and sulfur (SOx). By burning the coal in a fluidized bed of limestone (calcium carbonate) particles, the gaseous sulfur reacts with the limestone, allowing the sulfur to be removed in solid form rather dissipated into the atmosphere in gaseous form. In addition, the vigorous mixing of gas and solid particles in bubbling and circulating fluidized-beds enhances the combustion reaction and helps to eliminate anomalies in temperature or fuel distribution, which helps to lower the production of NOx emissions.
1.4 FLUIDIZED-BED BOILERS

In a boiler, the combustion of fuel in air generates energy that needs to be transferred to the water or steam; in this case, the fluidized bed is also a heat exchanger. Steam flows through tubes or pipes which come in contact with the fluidized bed. Typically, these tubes might actually be immersed in the bed, as in a BFB, or they might pass around the periphery of the riser, as in a CFB. The erosive nature of the high-velocity environment in a CFB generally dictates the need for placing these heat transfer surfaces on the periphery of the riser.

1.4.1 Steam-based power generation

Power generation with steam as the primary working fluid can be described by the Rankine cycle. Figure 1.4 shows a schematic for an ideal cycle.

![Schematic for an ideal Rankine cycle](image)

**FIGURE 1.4:** Schematic for an ideal Rankine cycle

In an ideal Rankine cycle, water is pumped to the boiler where it is converted to steam by interaction with a heat source (in this case, the combustion of fuel in air). The steam is then expanded through a steam turbine, and the energy extracted in the expansion process can be used to drive a generator. Following this, the steam passes through a condenser where another heat transfer interaction, this time with a heat sink, causes the steam to condense to water in the original state. Such a closed cycle is very idealized, but it is also illustrative of the use of steam for power generation. A real system will typically have, among other additions, a "scrubber" unit downstream of the exhaust of the combustion gases in order to suppress pollution. The use of
fluidized beds as boilers provides a distinct advantage, in that the need for additional exhaust clean-up can be eliminated.

1.4.1.1 Combined-cycle power generation
Alternative cycles that take advantage of the exhaust heat from gas side of the boiler can also be implemented to improve overall cycle efficiencies. "Combined cycle" refers to the use of both the steam and gas streams for power generation, and fluidized-bed boilers are equally advantageous in this situation. In this situation, the combustion process occurs at an elevated pressure, and the combustion products are passed through a gas turbine for power generation. Figure 1.5 gives a sample schematic for a combined cycle, where the gas stream ideally follows the Brayton-Joule cycle for power generation (isentropic compression and expansion, isobaric heat addition and rejection).

![Diagram of combined cycle](image)

FIGURE 1.5: Typical combined cycle configuration

Again, a fluidized-bed boiler can be operated equally effectively at elevated pressure in a combined cycle configuration as it can at atmospheric pressure in a simple cycle configuration. In general, though, boilers will be specifically designed for one configuration or the other.
1.4.2 Typical boiler design parameters
1.4.2.1 Riser size
As this discussion of fluidized beds moves along, it seems prudent to give some idea about the size and scale of the items mentioned. The CFB boiler in which most of the work in this study is a cold-flow scale-model of a 2 MWth unit; in other words, the full-sized model consumes fuel at a rate of 2 MW at peak capacity. The riser of this bed has a square cross-section of 0.64 m on a side, and it is 8 m tall. Other boilers are rated up to 100 MW or more; since the power level scales with the area of the bed, roughly, then a given linear dimension roughly scales by the square root of the increase in power. As a result, a large boiler might be several meters in diameter and perhaps nearly 50 m in height. Height-to-diameter ratios can range from about 10:1 to 50:1, and the height is driven by the design for the combustion and pollutant-absorption reactions (i.e., allowing enough time for the upward-traveling gas and particles to interact). Figure 1.6 presents a picture of a full-sized atmospheric CFB boiler with an energy output rate of 160 MWth; for comparison in size, notice the person in the lower left corner.

1.4.2.2 Gas properties
A typical simple-cycle boiler runs at atmospheric pressure, while a typical combined-cycle boiler might run at about 10-15 atmospheres pressure. The operating temperature in either configuration is typically 1000-1100 K, designed to stay below the activation temperature for NOx, which is about 1200 K. The gas superficial velocity will typically be around 5-10 m/s in a CFB. It is reasonable to model the properties of the combustion products as being similar to those of air at the same temperature and pressure, since both compounds are mostly nitrogen and can be shown to have similar thermodynamic properties [see Keenan, et al., 1992, e.g.].

1.4.2.3 Bed material properties
The bed material for a fluidized-bed boiler is typically a calcium-based solid, such as limestone. The specific gravity of the material is approximately 2.5, yielding a density of about 2500 kg/m³. A typical CFB boiler has particles that average 200-300 μm in diameter, and the distribution of sizes around this average can range from 50-500 μm. Another characterization of the solid particles is a parameter known as "sphericity", or the ratio of the volume occupied by the particle to the volume occupied by a sphere of the same effective diameter. The sphericity is basically a measure of the roundness of the particles, and one typically finds a sphericity of 0.6-0.9 for particles in fluidized-bed boilers. In a CFB boiler, the superficial velocity of the solid particles introduced in the bottom of the riser (mass flow rate of solids per unit area and solid density) will be a small fraction of the gas superficial velocity, typically 0.02%-0.2%. 

18
FIGURE 1.6: Full-sized CFB boiler, 160 MW\textsubscript{th} [from Kokko, et al., 1995]
1.4.2.4 Heat transfer in CFBs
Basu and Nag [1996] recently reviewed published data for heat transfer coefficients in CFB boilers. Most of the reported data is from laboratory-scale models running at low temperature, but some of the data is also from prototype-scale or full-sized commercial boilers. In low temperature models, heat transfer coefficients typically range from 50-250 W/m²-K with higher values typically observed for higher levels of solid concentration and shorter heat transfer surfaces. At full operating temperatures, the radiative heat transfer becomes appreciable and approximately equal to the rate of heat transfer without radiation; heat transfer coefficients up to 500 W/m²-K have been measured in full-scale CFB boilers.

1.4.3 Utilization of fluidized-bed boilers
In the past two decades, the popularity of fluidized beds as boilers has increased greatly, as seen in Figure 1.7. This growth trend has been driven by stricter government regulation of pollution from electric utilities and industrial plants, and both boiler manufacturers and their customers have taken advantage of the environmental benefits of fluidized beds.

![Graph showing annual installations of fluidized-bed boilers, worldwide, from 1973 to 1990.](image-url)

FIGURE 1.7: Annual installations of fluidized-bed boilers, worldwide [CIRO, 1990]
Fig. 1.7 also shows that there has been a marked shift from BFBs to CFBs in the past ten years; this is due largely to the perception that CFBs represent a better technology for exhaust gas cleanup. A CFB has a more dilute concentration of solid particles than a BFB, and the particles in a CFB are typically smaller and therefore have a larger surface-volume ratio. Both of these attributes provide for a better opportunity for better gas-solid contact and enhancement of the absorption of SO₂; in short, the CFB is typically a better chemical reactor [Squires, et al., 1985]. Furthermore, a CFB will typically occupy less square footage on the ground since it requires a higher gas velocity (i.e., less area for the same flow rate or power rating). CFBs are generally taller and more complex to operate, however, which is why some users still prefer BFBs.

1.4.4 Relevance of research in CFB boilers
Although the demand for CFB technology in boilers has been high, the application of CFBs as boilers is fairly recent. Given this, there has also been high demand for research into the physical phenomena underlying the technology, and the heat transfer processes in particular, in an effort to improve the design and operation of CFB boilers. Many of the early design efforts simply extended correlations from BFBs or extrapolated from the limited knowledge base of CFBs without regard to fundamental physical changes. The development of valid scale models for experimentation and analytical models [see Glicksman, et al., 1994] have contributed greatly to a better understanding of the physical mechanisms governing heat transfer in fluidized beds in general and in CFBs in particular.
1.5 REFERENCES


2. HYDRODYNAMICS AND HEAT TRANSFER IN CIRCULATING FLUIDIZED BEDS

2.1 A DESCRIPTION OF CFB HYDRODYNAMICS

2.1.1 Large-scale
A broad view of the flow of gas and solids in a CFB is shown in Figure 1.3. In a steady state, the air flow carries a certain mass of particles out the top of the riser; a solid conveyance system is in place to feed an equal mass flow of particles to the bottom of the riser. Typically, the particles flowing out the top are collected and recycled back to the bottom, and in a full-sized CFB boiler, the feed system also serves as a means for supplying chemically "fresh" particles. In the riser of a CFB operating in a steady state, then, there is a net upward flow of solid material.

2.1.2 Mid-scale
On closer inspection of the flow in the upper portion of a CFB riser, one typically observes a "core-annular" structure along the lines of that shown in Figure 2.1 [Yerushalmi & Cankurt, 1979, e.g.]. The core of the riser, perhaps the inner 80-90% of the cross-section, contains a relatively dilute and upward-flowing concentration of solid particles. The annular region, perhaps the outer 10-20% of the cross-section, contains a relatively dense and downward-flowing concentration of solid particles; still, the inlet and exit conditions of the riser indicate that the net flow of solid particles is upward.

![Core-annular flow structure, typical of CFBs](image)

**FIGURE 2.1:** Core-annular flow structure, typical of CFBs
In simple terms, the existence of the annular region can be attributed to the gas-phase boundary layer near the wall of the riser, in which a region of low gas velocity lacks the momentum to entrain the particles in that region. Solid material gets into the annular region via diffusion from the core region or from reflection from the top of the riser (i.e., failure to exit the riser). The constant diffusion of material to and from the core is sometimes referred to as the "renewal" of material in the annular region.

2.1.3 Small-scale
When considering heat transfer from CFBs, one focuses on the annular region. Heat transfer tubes are not typically immersed in the core of the bed because of the erosive nature of the high-velocity gas and solid flow. Instead, the tubes are embedded in the riser wall, thereby removing them from a harsh environment and exposing them only to the annular region of the flow in the riser. In this annular region, the downward flow of solid material is not continuous and actually occurs intermittently, with the solid particles flowing in agglomerations called "strands" or "clusters" [see Yerushalmi & Cankurt, 1979, or Ishii, et al., 1989, for example]. Typical cluster formations are shown in Figure 2.2.

![Flow of clusters](image)

**FIGURE 2.2:** View at CFB wall, showing clusters of particles

Brereton and Grace [1993] developed an experimental parameter called the "intermittency index"; as this index approaches zero, the nature of the flow approaches the core-annular description. They found relatively low values to exist in the upper regions of a CFB riser.
To understand heat transfer in a CFB, then, it becomes clear that an understanding of the nature of the clusters, their contact with the heat transfer surfaces, and the flow in the annulus is required. The physical characteristics of the clusters – size, shape, speed, density – as well as the characteristics of the annular flow – annulus thickness, concentration of clusters, renewal rate, separation from the wall – are highly dependent upon the operating conditions in the CFB riser.
<table>
<thead>
<tr>
<th>Investigators</th>
<th>Bed dim.</th>
<th>Htd. lgth.</th>
<th>Temp.</th>
<th>Mat'l.</th>
<th>Part. dia.</th>
<th>h (W/sq.m-K)</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kobro &amp; Brereton</td>
<td>n/a</td>
<td>n/a</td>
<td>25-850°C</td>
<td>sand</td>
<td>170 µm</td>
<td>80-300</td>
<td>higher h with smaller part. size, higher solid fraction, higher bed temp.</td>
</tr>
<tr>
<td>[1986]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>250 µm</td>
<td>50-200</td>
<td></td>
</tr>
<tr>
<td>Subbarao &amp; Basu</td>
<td>4.5 m tall</td>
<td>0.10 m dia.</td>
<td>0.10 m</td>
<td>room</td>
<td>sand</td>
<td>130/260 µm</td>
<td>higher h with higher solid fraction; no trend observed with part. size, but op. conditions unclear</td>
</tr>
<tr>
<td>[1986a]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>150-300</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Basu &amp; Nag</td>
<td>5.5 m tall</td>
<td>0.10 m dia.</td>
<td>0.10 m</td>
<td>room</td>
<td>sand</td>
<td>87/227 µm</td>
<td>higher h with higher solid fraction, smaller part. size; generally higher h on short surface</td>
</tr>
<tr>
<td>[1987]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>150-300</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wu, et al. [1987]</td>
<td>7.3 m tall</td>
<td>0.15 m sq.</td>
<td>1.53 m</td>
<td>170-400°C</td>
<td>sand</td>
<td>188/356 µm</td>
<td>higher h with higher solid fraction, somewhat with smaller part. size; little dependence on gas vel.</td>
</tr>
<tr>
<td>Basu &amp; Konuche</td>
<td>6.7 m tall</td>
<td>0.20 m sq.</td>
<td>0.20 m</td>
<td>650-885°C</td>
<td>sand</td>
<td>296 µm</td>
<td>higher h with higher solid fraction and bed temperature (increase in radiative transfer)</td>
</tr>
<tr>
<td>[1988]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>50-200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Furchi, et al.</td>
<td>6.0 m tall</td>
<td>n/a</td>
<td>250°C</td>
<td>glass</td>
<td>109/196/269 µm</td>
<td>30-150</td>
<td>higher h with higher solid fraction, smaller part. size; no trend observed with gas velocity</td>
</tr>
<tr>
<td>[1988]</td>
<td>0.07 m dia.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wu, et al. [1989]</td>
<td>7.3 m tall</td>
<td>0.15 m sq.</td>
<td>1.59 m</td>
<td>340-880°C</td>
<td>sand/lime mix.</td>
<td>220-240</td>
<td>higher h with higher bed temp., solid fraction; thermal development region observed at top of surface</td>
</tr>
<tr>
<td>Basu [1990]</td>
<td>height √/a</td>
<td>0.10 m dia.</td>
<td>0.10 m</td>
<td>room</td>
<td>sand</td>
<td>87-250</td>
<td>higher h on shorter surface; higher h with smaller particle size</td>
</tr>
<tr>
<td>[1990]</td>
<td>0.10 m dia.</td>
<td></td>
<td></td>
<td></td>
<td>60-350</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Basu [1990]</td>
<td>6.3 m tall</td>
<td>0.20 m sq.</td>
<td>0.80 m</td>
<td>600-900°C</td>
<td>sand</td>
<td>296 µm</td>
<td>higher h with higher bed temp.; lower h on longer surface</td>
</tr>
<tr>
<td>Bi, et al. [1991]</td>
<td>8.0 m tall</td>
<td>0.19 m dia.</td>
<td>0.04-0.16 m</td>
<td>room</td>
<td>FCC</td>
<td>48 µm</td>
<td>higher h on shorter surface; higher h with smaller particle size</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>200-600</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**TABLE 2.1:** Review of recent measurements of heat transfer in CFBs
<table>
<thead>
<tr>
<th>Investigators</th>
<th>Bed dim.</th>
<th>Htd. lgth.</th>
<th>Temp.</th>
<th>Mat'l.</th>
<th>Part. dia.</th>
<th>$h$ (W/sq.m-K)</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wu, et al. [1991]</td>
<td>9.3 m tall 0.15 m dia.</td>
<td>0.01 m</td>
<td>room</td>
<td>sand</td>
<td>171 μm</td>
<td>150-250 ±50 fluctuating</td>
<td>instantaneous measurements as high as 300 W/sq.m-K; average $h$, min/max. increase with solid fraction</td>
</tr>
<tr>
<td>Chen &amp; Chen [1992]</td>
<td>6.0 m 0.09 m dia.</td>
<td>0.12 m</td>
<td>n/a</td>
<td>FCC</td>
<td>71 μm</td>
<td>200-600</td>
<td>higher $h$ on shorter surface; higher $h$ with higher solid fraction</td>
</tr>
<tr>
<td>Dou, et al. [1992]</td>
<td>11 m tall 0.15 m dia.</td>
<td>0.006 m</td>
<td>room</td>
<td>sand</td>
<td>125 μm</td>
<td>100-350</td>
<td>high $h$ for instantaneous measurements</td>
</tr>
<tr>
<td>Leckner &amp; Andersson [1992]</td>
<td>13.5 m tall 1.7 m sq.</td>
<td>11.5 m 850°C</td>
<td>lime</td>
<td>≈250 μm</td>
<td>50-300</td>
<td>higher $h$ with solid fraction; high temp. yields higher max. $h$ even on long surface</td>
<td></td>
</tr>
<tr>
<td>Burki, et al. [1993]</td>
<td>11 m tall 0.15 m dia.</td>
<td>0.92 m</td>
<td>room</td>
<td>sand catalyst</td>
<td>245 μm 97 μm</td>
<td>≈20-50 ≈20-70</td>
<td>thermal development regions observed at both top and bottom of surface (particle and gas layers)</td>
</tr>
<tr>
<td>Ebert, et al. [1993]</td>
<td>8.2 m tall 0.20 m dia.</td>
<td>0.14 m</td>
<td>room</td>
<td>glass</td>
<td>88 μm</td>
<td>150-300</td>
<td>comparable mass transfer experiments indicate substantial contribution to $h$ from particles</td>
</tr>
<tr>
<td>Hyre &amp; Glicksman [1995]</td>
<td>0.46 m tall 0.04 m sq.</td>
<td>0.46 m</td>
<td>room</td>
<td>glass</td>
<td>117 μm</td>
<td>80-120</td>
<td>very small bed; showed ≈30% enhancement with small lateral ridges</td>
</tr>
<tr>
<td>Lockhart, et al. [1995]</td>
<td>9.3 m tall 0.15 m dia.</td>
<td>0.91 m</td>
<td>room</td>
<td>sand</td>
<td>200 μm</td>
<td>100-250</td>
<td>measured lateral variation on membrane wall; higher $h$ observed on fin vs. crest</td>
</tr>
<tr>
<td>Andersson [1996]</td>
<td>13.5 m tall 1.7 m sq.</td>
<td>11.5 m 850°C</td>
<td>sand</td>
<td>220/300/400 μm</td>
<td>50-200</td>
<td>local $h$ increases with smaller particle size; long surface smooths out variations in avg. $h$</td>
<td></td>
</tr>
<tr>
<td>Basu, et al. [1996]</td>
<td>2.02 m tall 0.05 m dia.</td>
<td>0.15 m 250-750°C</td>
<td>sand</td>
<td>232/507 μm</td>
<td>70-200</td>
<td>higher $h$ with higher solid fraction, smaller part. size, higher pressure, higher temp.</td>
<td></td>
</tr>
<tr>
<td>Wang, et al. [1996]</td>
<td>6.0 m tall 0.15 m dia.</td>
<td>0.06 m room</td>
<td>alumina</td>
<td>75 μm</td>
<td>200-400</td>
<td>relatively high $h$ (convection only in cold model) on short heated surface</td>
<td></td>
</tr>
</tbody>
</table>

TABLE 2.1: Review of recent measurements of heat transfer in CFBs (cont'd.)
2.2 HEAT TRANSFER IN CFBS

2.2.1 Measurements of heat transfer in CFBS

Over the last decade or so, coincident with the level of interest in CFBS, there have been a variety of studies performed in the interest of better understanding heat transfer at the wall of a CFB riser. Basu and Nag [1996] recently conducted a review of the work and the general results, and Table 2.1 presents the results of the author's own review. The general results and trends observed are quite similar. Higher heat transfer coefficients are generally observed in CFBS running at a higher temperature (more radiative transfer), with higher concentration of solid particles (attributed to an increase in conduction from the particles), with smaller particles (attributed to lower thermal inertia for each particle), with shorter surfaces (still developing thermally) and with system pressure. The bed-to-wall heat transfer rates also appear to be independent of the superficial gas velocity, and this is attributed to the fact that the gas phase does not typically cover much of the heat transfer surface. It is difficult to determine the effect of bed height or diameter on heat transfer since the operating conditions vary from case to case, although Glicksman [1997] has compiled some data showing an increase in heat transfer with bed diameter.

2.2.2 Modeling heat transfer in CFBS

Based on the flow phenomena described previously, a mechanistic model for heat transfer in a CFB can be and has been developed. This model accounts for heat transfer at the wall from both the gas (dilute) phase and the solid (dense) phase given the typical flow phenomena, and it is discussed in more detail in the next section. The field of fluidization dates back 50 years or more, so that much of the theoretical work has been established for understanding heat transfer in fluidized beds. This model neglects the effects of radiative heat transfer, however, by assuming that convection and radiation are independent and additive. The validity of this assumption is discussed later. Similar models have been presented by Basu [1990] and Wu, et al. [1990], among others.

Other models have been proposed [see Martin, 1981, Molerus, 1993, or Wirth, 1995, for example], but it is the author's opinion that the present model is the least obtuse and the most physically intuitive. Martin's model, for example, considers a contribution from the dense phase coming from individual particles, each with their own independent motion. Although this model matches some heat transfer data, it does not physically represent the observed flow behavior. The model of Molerus and Wirth is semi-empirical in nature and has other shortcomings that are discussed later.
2.2.3 Development of the heat transfer model

2.2.3.1 Gas and particle convection

Since both the gas and the particle clusters flow independently along the heat transfer surface, one can consider that they contribute independently to convective heat transfer (where convection is defined as conduction in a moving medium). This concept was first proposed by Subbarao & Basu [1986] and it involves the definition of gas- and particle-convective heat transfer coefficients, as well as a hydrodynamic parameter describing the fraction of the heat transfer surface covered by clusters of particles:

\[ h_{\text{convective}} = f \cdot h_{pc} + (1 - f) \cdot h_{gc} \]  \hspace{1cm} (2.1)

In Eq. (2.1), \( f \) represents the fractional coverage by clusters, \( h_{pc} \) represents the particle-convective contribution and \( h_{gc} \) represents the gas-convective contribution. Determining the particle-convective contribution requires more analysis; as discussed later, \( h_{gc} \) can be approximated by single-phase heat transfer correlations for forced convection. The gas-convective heat transfer is typically a small portion of the total convective heat transfer except when the concentration of particles in the riser is relatively low.

2.2.3.2 Particle convection – cluster-wall contact

Given the renewal mechanisms between the core and annulus in a CFB, the cluster-wall interaction is likely to be of short duration so that the heat transfer can be analyzed as an unsteady process in each cluster. Furthermore, it is postulated that this interaction is short enough, or that the clusters are large enough, that a cluster can be treated as a semi-infinite body. This results in the following expression for heat transfer from a cluster:

\[ h_{\text{cluster}} = \sqrt{\frac{(k \rho c)_{\text{cluster}}}{\pi t}} \]  \hspace{1cm} (2.2)

This part of the model was first developed by Mickley and Fairbanks [1955] for bubbling fluidized beds, in which the semi-infinite body model was proposed for densely-packed particles passing across immersed tubes. Eq. (2.2) also shows the dependence of heat transfer on hydrodynamic parameters: the cluster-wall contact time, \( t \), and the cluster properties \( k, \rho \) and \( c \), through their dependence on the concentration of solid particles in the cluster.
### 2.2.3.3 Particle convection – cluster properties

As seen in Eq. (2.2), the heat transfer from a cluster depends on some of its physical properties: thermal conductivity, $k$, and volumetric heat capacity, $(\rho c)$. Assuming that the concentration of solid particles is fairly uniform throughout the cluster, the volumetric heat capacity can be calculated for a mixture of a given mass of air and a given mass of solid particles:

$$\left(\rho c\right)_{\text{cluster}} = \varepsilon_{\text{cluster}} (\rho c)_{\text{air}} + (1 - \varepsilon_{\text{cluster}}) (\rho c)_{\text{solid}}$$  \hspace{1cm} (2.3)

where $\varepsilon_{\text{cluster}}$ represents the concentration of air (void) by volume of a cluster. Since $(\rho c)_{\text{air}} \ll (\rho c)_{\text{solid}}$, as long as $\varepsilon_{\text{cluster}} < 0.99$ it can be shown that Eq. (2.3) reduces to:

$$\left(\rho c\right)_{\text{cluster}} = (1 - \varepsilon_{\text{cluster}}) (\rho c)_{\text{solid}}$$  \hspace{1cm} (2.4)

Determining the thermal conductivity of a cluster is much more complex, considering the multiple heat conduction paths between neighboring particles. Fortunately, though, Gelperin and Einstein [1971] provide a formula to represent empirical data for $k_{\text{cluster}}$ as a function of $k_{\text{solid}}$, $k_{\text{gas}}$ and $\varepsilon_{\text{cluster}}$:

$$\frac{k_{\text{cluster}}}{k_{\text{gas}}} = 1 + \frac{(1 - \varepsilon_{\text{cluster}}) \cdot (1 - k^{*})}{k^{*} + 0.28 \varepsilon_{\text{cluster}}^{0.63} k_{\text{gas}}^{0.18}}$$  \hspace{1cm} (2.5)

In Eq. (2.5), $k^{*}$ represents the ratio of $k_{\text{gas}}$ to $k_{\text{solid}}$, and it can be shown that $k_{\text{cluster}}$ is about 1.5 to 2 times $k_{\text{gas}}$ for typical CFB conditions. In general, Eqs. (2.4) and (2.5) quantify the dependence of heat transfer from a cluster to the concentration of particles in a cluster, where $\varepsilon_{\text{cluster}}$ is a parameter that is dependent upon the operating conditions in the CFB.

### 2.2.3.4 Particle convection – contact resistance

Investigation of Eq. (2.2) shows that as the cluster-wall contact time approaches zero, the rate of heat transfer from the cluster approaches infinity. This is both physically impossible and contrary to observations. In fact, a limiting rate of heat transfer is typically observed, and this can be explained by using the concept of a contact resistance between the cluster and the wall. This concept was first proposed by Baskakov [1964], in which the imperfect contact between the particles and the wall – either due to a certain gap between the solid phase and the wall or due to asperities in the surface of the solid phase – results in a thermal resistance. The analytical solution for heat transfer from a semi-infinite body to a constant-temperature wall via a constant contact resistance is given by:

$$h_{pc} = \frac{k_{\text{gas}}}{y_{gap}} \cdot \exp \left[ \frac{\left( k_{\text{gas}} \right)}{y_{gap}} \cdot \frac{1}{(k \rho c)_{\text{cluster}}} \right] \cdot \text{erfc} \left[ \frac{\left( k_{\text{gas}} \right)}{y_{gap}} \cdot \frac{1}{(k \rho c)_{\text{cluster}}} \right]^{1/2}$$  \hspace{1cm} (2.6)
where $y_{gap}$ represents the effective size of the gap between the cluster and the wall, and "erfc" is the complementary Gaussian error function. Eq. (2.6) is difficult to work with analytically, and it can be shown that the particle-convective heat transfer can be well approximated by adding the simple semi-infinite body solution of Eq. (2.2) to the steady-state contact resistance of a gap described by $y_{gap}$ as if they were both steady-state thermal resistances connected in series:

$$h_{pc} = \frac{y_{gap}}{k_{gas}} + \sqrt{\frac{\pi t}{k \rho c}}_{\text{cluster}}^{-1}$$  \hspace{1cm} (2.7)

This equation is in a more convenient and intuitive form than Eq. (2.6).

2.2.3.5 Model summary

Substituting Eq. (2.7) into Eq. (2.1) yields the following expression for convective heat transfer at the wall in a CFB:

$$h_{\text{convective}} = f \cdot \left( \frac{y_{gap}}{k_{gas}} + \sqrt{\frac{\pi t}{k \rho c}}_{\text{cluster}}^{-1} \right) + (1 - f) \cdot h_{gc}$$  \hspace{1cm} (2.8)

From Eq. (2.8), one sees that convective heat transfer in a CFB is really a function of the hydrodynamics of the gas-solid flow. Mechanistically, it has been shown that the rate of convective heat transfer depends on $f$, $y_{gap}$, $t$, $e_{\text{cluster}}$, and $h_{gc}$, all of which depend upon operating conditions in the CFB.

The existence of particle convection has been called into question by Molerus [1993] and Wirth [1995], for example, in which they postulate mechanisms for heat transfer first from the individual particles to the gas and then from the gas to the wall. However, the significance of particle convection has been shown experimentally by Ebert, et al. [1993]. A typical convective heat transfer coefficient in a CFB is about 200 W/m²-K; for single-phase gas flow under the same conditions, it might be about 50 W/m²-K. The observed increase could be attributed to simple enhancement of mixing in the fluid boundary layer due to the presence of particles. In the experiments of Ebert, however, both heat and mass transfer rates were measured in a CFB riser with air flowing and without particles. The mass transfer augmentation with the particles present was approximately 50% and could be attributed solely to mixing enhancement, since the particles were inert in the experiments. The heat transfer augmentation, on the other hand, was more on the order of 500%, indicating the presence of an additional transport mechanism due to the presence of the particles. The solid particles in Ebert's experiments were glass beads with an average diameter of 88 μm; for such small particles, Wirth's model ignores the particle-convection mechanism.
Finally, it should be noted that radiative transfer has been neglected. Until recently, it was assumed that radiative heat transfer could be treated separately and considered additive to the convective heat transfer. In this type of analysis, it is assumed that the face of a cluster radiates directly to the wall in the area covered by clusters and the core of the bed radiates to the wall in the area uncovered by clusters. Typically, the temperature at the face of the cluster is taken to be that calculated due to cooling according to Eq. (2.7). One modification to this is to incorporate radiative transfer from the cluster surface to the wall in order to calculate a surface temperature [Fang, et al., 1995b]. Recent analyses, however, indicate that radiation from the core of the bed incident on the back of a cluster will actually pass through, somewhat attenuated, and this radiant flux will also distort the temperature profile in the cluster, thereby invalidating the semi-infinite body analysis [see Flamant, et al., 1994, and Farrell & Glicksman, 1997]. These analyses have yet to be validated experimentally, though, and the CFB heat transfer model developed here is certainly applicable to low-temperature applications in which radiative transfer is truly negligible.

2.2.4 Review of prior research

In the last ten years or so, a variety of researchers have sought to measure the hydrodynamic parameters in the heat transfer model of Eq. (2.8). There have been varying levels of success, and many of the results have already been compiled by Lints [1992]. A brief synopsis of these and more recent results will be provided in the following paragraphs.

2.2.4.1 Gas convection, fractional wall coverage

The gas-convective contribution to heat transfer in a CFB, as defined in Eq. (2.8), has been shown to be reasonably well approximated by correlations for single-phase gas convection. In other words, the standard form of $Nu_D = fn(Re_D, Pr)$ works well when the gas superficial velocity, the bed diameter and the gas-phase properties are used. The simplicity of this was shown by Lints and Glicksman [1993] in their analysis of data from a variety of sources.

The fraction of the wall covered by clusters is a slightly more complex phenomenon. In correlating various measurements, Lints and Glicksman [1993] showed that $f$ is dependent upon the average solid concentration in the cross-section, which is certainly a sensible conclusion. The observation that increasing bed diameter can increase heat transfer for the same level of solid concentration [see Glicksman, 1997] led Noymer, et al. [1995] to investigate the effect of bed diameter on fractional wall coverage, where larger values of $f$ were observed in larger beds. Given that $f = 1$ is a limiting condition, Hyre [1995] analyzed the physics behind solid deposition at the wall and fit the following function to seven sets of existing data:

32
\[ f = 1 - \exp\left[-6 \cdot \frac{u_*}{\rho_*^3} F r D^{-1/3} (1 - \epsilon)_{\text{avg}}\right] \] 

(2.9)

where \((1 - \epsilon)_{\text{avg}}\) is the average cross-sectional solid concentration and \(0 \leq f \leq 1\). The other parameters in Eq. (2.9) are dimensionless groups describing the operation of the bed and their definitions are discussed later.

2.2.4.2 Cluster-wall contact time

The contact time between a cluster and the wall, along with the contact gap between the cluster and the wall and the fractional wall coverage, is arguably one of the most fundamental of the hydrodynamic parameters in the particle-convective portion of the CFB heat transfer model in Eq. (2.8). At the same time, though, it is perhaps the least well-understood. Local cluster velocities have been measured previously [see Ishii, et al., 1989, and Rhodes, et al., 1992, e.g.] and there have been reports of indirect measurements of the length of contact between clusters and the wall [Wu, et al., 1991, and Fang, et al., 1995a]. Typically, one might take the length measurement and divide by the velocity to calculate a contact time. Using this method, one might find a contact time around one second, to an order of magnitude. However, there is no indication that the cluster velocity is constant for the entire contact length; in fact, it has been postulated that there is some acceleration to a terminal velocity, as with a body in gravitational free-fall [Glicksman, 1988]. The length measurements reported either measure the contact length by cross-correlating heat transfer rates from probes that are relatively far apart [Wu, et al., 1991] or they are inferred from heat transfer measurements by using the particle-convective heat-transfer model given in Eq. (2.8) [Fang, et al., 1995a]. In another approach, Chen and Chen [1992] inferred cluster-wall contact times from velocity measurements along a 12 cm section of the riser wall; this, however, is not necessarily representative of the full wall residence time. Since all of these results lack utility for the observation and computation of the cluster-wall contact time, it is apparent that a more direct method is needed for measurement.

Inspection of Eq. (2.8) shows that if the cluster-wall contact time can be shortened, for all else remaining unchanged, then the heat transfer rate should increase. The inverse dependence of heat transfer on contact time reflects the fact that the driving temperature difference drops as the cluster and the wall stay in contact. In the limit of infinitesimally short contact times, though, the resistance of the air gap dominates the particle-convective heat transfer. Ribbed surfaces are typically used in single-phase heat transfer systems in order to disturb and renew the fluid boundary layer, since the thickness of the boundary layer creates a resistance to heat transfer. Lateral ribs can have a similar effect on particle convection in a fluidized bed, in which the ribs cause the premature ejection of clusters from the heat transfer surface. If there is sufficient
diffusion of material to and from the wall, then the fraction of the wall covered by clusters can remain unchanged and heat transfer rates can increase. This phenomenon was observed experimentally by Hyre and Glicksman [1995], in which small ribs (≈ 150 μm) were shown to increase heat transfer beyond that predicted by simply including wall roughness in single-phase heat transfer correlations. Additionally, at some point after moving the ribs closer together, no further increase in heat transfer was observed. This is consistent with the dominance of the cluster-wall contact resistance for very short residence times as seen in Eq. (2.8).

2.2.4.3 Cluster solid concentration
A simple relation for determining $\varepsilon_{\text{cluster}}$ was developed by Lints and Glicksman [1993]. Their correlation of their experimental results and those of others yielded the following expression:

$$(1 - \varepsilon_{\text{cluster}}) = (1 - \varepsilon)^{1/2}$$

(2.10)

In other words, the solid concentration of clusters can be approximated by taking the square-root of the average cross-sectional solid concentration. As an example, if the average solid fraction is 1% or 2% by volume, as might typically be the case, then Eq. (2.10) will predict the solid fraction of the clusters to be about 10% to 15% by volume. More recent results and models yield similar values, albeit with no better physical understanding of what governs the solid concentration in a cluster. For example, Soong, et al. [1993] used a capacitance probe and measured the solid fraction of clusters between 10% and 30%, while Lockhart, et al. [1995] used a similar technique to find concentrations of 15% to 25%.

2.2.4.4 Cluster-wall contact gap
Based on experimental results in a scale-model CFB, Lints and Glicksman [1993] arrived at a correlation for the gap between moving clusters and the wall in which the only variable was the average cross-sectional solid concentration. As with $f$, it is hard to believe that $y_{\text{gap}}$ depends only on one parameter. In fact, inspection of their data shows a dependence on gas velocity: for the same average solid concentration, $y_{\text{gap}}$ is consistently higher for higher values of superficial velocity. This indicates that further analysis and experimentation is warranted for a better understanding of this phenomenon. To an order of magnitude, though, the size of the gap is equal to about one particle diameter (from 50-500 μm, depending on operating conditions).
2.2.4.5 Cluster shape and velocity

Although not explicitly part of Eq. (2.8), the shape and descent velocity of a cluster are characteristics of general interest that affect the motion of the cluster and therefore affect the heat transfer at the wall. There is no uniform method for measuring or reporting the shapes of clusters observed in CFBs. Lim, et al. [1996] described clusters as being "arch-shaped" or approximated by concentric half-ellipses. Horio and Kuroki [1994] made similar observations, but also noticed a distinct "tail" on the "paraboloid". It is the opinion of the author, based on personal observations, that both reports are correct; in fact, the shapes of clusters seem to vary from cluster to cluster in one CFB as much as they do between different CFBs.

There is a wide body of data available regarding the velocities of clusters. This characteristic seems to be the simplest to measure of all the hydrodynamic parameters discussed here; a video camera, a timer, and a tape measure are all that are required. The most noticeable aspect of the data is that the cluster descent velocities show little variation between different CFBs and operating conditions – observations put cluster velocities in the range of 0.5-2.5 m/s [see Ishii, et al., 1989, and Rhodes, et al., 1992, e.g.]. There is more discussion of cluster velocities later.

2.2.5 Summary of heat transfer modeling

The mechanistic model for convective heat transfer in a CFB, given by Eq. (2.8), depends almost solely on the small-scale hydrodynamic behavior. Ideally, Eq. (2.8) could be used in the design process for a CFB boiler as a method for predicting rates of heat transfer. Given the complexity of the gas-solid flow, however, models for the small-scale hydrodynamics are extremely difficult to develop from first principles. In-depth experimental study of the flow behavior is required to come to a better understanding of the mechanisms governing heat transfer, and from there, perhaps, a predictive heat transfer model can be developed.
2.3 SCALE MODELING OF CFBS

The ability to construct a scale-model CFB that exhibits behavior similar to a full-sized CFB is important for a variety of reasons. For example, many boiler manufacturers rely on data from smaller prototypes when developing a product, and the ability to scale up the prototype while maintaining control over desirable characteristics is essential. The use of prototypes is also a practical matter, saving time and expense relative to the construction of a full-sized CFB, especially when the specifications of the final product may be undefined. Finally, the use of a cold-flow (i.e., no combustion) laboratory model can be beneficial in that the lower-temperature environment is less severe and therefore more conducive to the use of specialized data-acquisition equipment.

The small-scale hydrodynamic parameters discussed in conjunction with modeling convective heat transfer in CFBS are perfect examples of characteristics that need unique equipment for measurement. In order to apply any results to a full-sized CFB, though, one must consider the proper methods for extrapolation or scale-up. For these reasons, scale modeling of fluidized beds has been a subject of much interest, and some standard rules have been developed [see Glicksman, et al., 1994, e.g.]. The methods for scale-modeling and some typical results are discussed next.

2.3.1 Scaling methods

In general, the technique that can be used to develop guidelines that ensure similitude in scale modeling is called "dimensional analysis". In dimensional analysis, one identifies all of the independent parameters important to the system and then systematically combines them into a number of dimensionless parameters. If there are \( n \) independent parameters consisting of \( r \) basic physical dimensions (length, mass, time, temperature) then there will be \( n - r \) independent dimensionless groups. In reducing the number of independent parameters, the number of experiments required to understand the behavior of a system is also reduced. It should be noted that the independent parameters are only those that can be controlled directly by design.

One can be more rigorous in dimensional analysis by considering all of the pertinent governing equations, boundary conditions, and constitutive relations. Similar to the simpler dimensional analysis, though, once the equations are written, all variables are non-dimensionalized using \( r \) appropriate dimensional variables. Algebraic manipulation of the subsequent equations yields the original governing equations in terms of dimensionless variables and dimensionless coefficients. If performed correctly, the number of dimensionless coefficients should be the same as those obtained using the simpler method. Many introductory textbooks on fluid mechanics contain a more complete description of these types of analyses; one reference is the book by Fox & McDonald [1985].
2.3.2 Application to fluidized beds

To properly simulate the flow in a CFB, one must first consider the independent parameters that are important from a hydrodynamic standpoint. Given the vigorous mixing promoted by the presence of solid particles in a CFB, most of the gas flow in the riser can be represented by exhaust products at a certain temperature and pressure – or since only two variables are required to specify a thermodynamic state, they can be represented by a certain density and viscosity ($\rho_f$ and $\mu$). In this way, temperature and pressure-dependence are removed from the hydrodynamic scaling. Certainly the dimensions of the riser ($L$ and $D$) are important, as are the average size and density of the particles ($d_p$ and $\rho_s$). The gas flow rate is characterized by the gas superficial velocity ($u_0$) and the flow rate of solid particles is often expressed as a flux or flow rate per unit riser area ($G_s$). Finally, given the vertical nature of the flow, gravity plays a role, so $g$ should be included among the independent parameters. In addition to these parameters, there may be others that are important: parameters describing particle geometry, such as the size distribution of the particles and their sphericity; and parameters describing inter-particle interactions, such as collisional and electrostatic interactions.

Nine independent parameters have been identified as being important for hydrodynamic similitude, and among them, they contain three independent physical dimensions – mass, length and time. Therefore, one parameter needs to be selected to represent each dimension and the three are then combined with each of the remaining six parameters to produce the following six dimensionless groups that must be matched for similarity:

$$\frac{L}{D}, \frac{u_0^2}{gD}, \frac{\rho_s}{\rho_f}, \frac{d_p}{D}, \frac{\rho_f u_0 D}{\mu}, \frac{G_s}{\rho_s u_0}$$

(2.11)

Previous work by Nicastro and Glicksman [1984] and Westphalen and Glicksman [1993], among others, has shown that matching the parameters in Eq. (2.11) between a full-sized bed and a cold-flow scale-model is sufficient to match the overall hydrodynamic conditions on a dimensionless basis. They also found that other parameters such as particle size distribution and sphericity are important, while parameters relating to electrostatic or collisional interactions are not as important.

In the typical design of a cold-flow scale model, one would like to specify the gas to be used, its temperature and pressure, as well as the linear scale factor for the CFB. However, the parameters of Eq. (2.11) are restrictive; working through the design of scale model shows that both the state of the gas and the size of the bed cannot be specified independently. This presents a problem when attempting to construct scale models of fluidized beds running at about 10 atm pressure, in which
the cold-flow model must actually be built to roughly the same size as the full-sized CFB. In this case, it becomes beneficial to make simplifications to the scaling parameters.

2.3.3 Simplification of the scaling laws
Reducing the number of group in Eq. (2.11) requires inspection of the governing equations and constitutive relations describing the flow of the gas and solid. The simplification analysis was performed by Glicksman, et al. [1993] for both high and low values of $Re$ based on gas superficial velocity and particle diameter. This analysis resulted in the following dimensionless groups:

$$
\frac{L}{D}, \frac{u_2^2}{g D}, \frac{\rho_s}{\rho_f}, \frac{u_0}{u_{mf}}, \frac{G_s}{\rho_s u_0} \tag{2.12}
$$

where $u_{mf}$ is the minimum fluidization velocity, or the gas superficial velocity at which the momentum loss through the bed is equal to the weight of the bed [see Grace, 1982, e.g.]. Using slightly different notation, the groups in Eq. (2.12) can be expressed as:

$$
L^*, Fr_D, \rho^*, u^*, G^* \tag{2.13}
$$

where the asterisk subscript denotes a dimensionless group, and $Fr_D$ is the Froude number based on the diameter of the riser. As a result of the simplification, the additional degree of freedom allows for the linear scale factor of the fluidized bed to be specified in addition to specifying the gas and its properties. Experimental validation of this simplified set of scaling parameters was also performed for linear scale factors as low as 1/16 for atmospheric beds [Glicksman, et al., 1993] and 1/6.5 for pressurized beds [Hyre, et al., 1997]. Similar to the full set of scaling laws, the particle size distribution and sphericity are considered important, while other inter-particle properties are not. Application of the groups in Eq. (2.13) to a cold scale-model of an atmospheric-pressure CFB results in the need for particulate materials that are denser and finer than those found in the full-sized CFB; an example of this is given in the next chapter.

2.3.4 Relevance of scale modeling to heat transfer
Scale modeling can also be applied to heat transfer, in which a thermally similar models can be constructed. In this case, one would introduce additional parameters such as the thermal conductivity of the gas and the solid and the specific heat of the solid, and this would result in additional dimensionless groups. As seen before, though, the six dimensionless groups of Eq. (2.11) are already restrictive, so that adding more dimensionless groups will make it difficult or impossible to construct a scale-model that is both thermally and hydrodynamically similar.
Recognizing that the total convective heat transfer is governed largely by hydrodynamic characteristics of the gas-solid flow, a thermally similar model need not be constructed in order to
gain more insight into heat transfer. One sees from Eq. (2.8) that if parameters such as \( f \), \( \varepsilon_{\text{cluster}} \), \( y_{\text{gap}} \) and \( t \) can be measured, then a convective heat transfer coefficient can be predicted as long as \( k \) and \( c \) are known. Neither the thermal conductivity nor the specific heat capacity of the solid particles are expected to affect the hydrodynamic characteristics in the CFB.
2.4 THESIS PROJECT GOALS

Given that the model for particle convection is dependent on several hydrodynamic parameters, it would be beneficial to observe, analyze, and understand these phenomena in order to allow the heat transfer model to be predictive. This would facilitate the thermal design of CFB combustors, and it would provide a basis for making rational improvements in the design of CFBs. Many of the hydrodynamic parameters involved have already been investigated, as discussed earlier, but measurement of the contact time of clusters at the wall of a CFB has proved difficult. Accordingly, the primary goals of this project are to: develop a technique to measure $t$, the cluster-wall contact time; employ this technique to investigate the variation of $t$ with gas superficial velocity, solid recycle rate, solid density and particle size; investigate $t$ for flat walls, to provide a general basis for understanding the phenomenon, and for a wall with longitudinal ribs, to simulate the water tubes in the walls of full-scale CFBs; supplement the hydrodynamic data with an analytical model or models to explain the observed behavior and predict trends for other operating conditions. The remainder of this thesis is organized along the lines described above.
2.5 REFERENCES


K.S. Lim, J. Zhou, C. Finley, J.R. Grace, C.J. Lim and C.M.H. Brereton, Cluster descending velocity at the wall of circulating fluidized bed risers, 5th International Conference on Circulating Fluidized Beds, Beijing, PRC (1996).


3. EXPERIMENTAL APPARATUS AND METHODS

3.1 OVERVIEW
The scale-model CFB used in this project was originally built as a 1/4-scale model of a combustor with a fuel rating of 2 MW [Westphalen, 1993]; the relatively small heat rate is indicative of the fact that the "full-sized" combustor is actually a pilot or prototype CFB. Figure 3.1 shows a general schematic of the entire system, and some of the design information for the system is contained in Appendices A and B.

![Diagram of experimental CFB](image)

**FIGURE 3.1: Schematic of experimental CFB**
3.2 SPECIFICATIONS FOR THE SCALE-MODEL CFB

3.2.1 General system construction

As shown in Fig. 3.1, the CFB system consists of a few basic components: an air supply system ("blower"), a main flow chamber ("riser"), a particle separation system ("separator"), a particle recycling system ("downcomer"), and an air filtration and exhaust system.

3.2.1.1 Air supply

Air is supplied to the CFB from a blower that is part of the central facility. The volumetric flow capacity of the blower is about 0.5 m³/s; a flow rate of only about 0.1 m³/s is required for these experiments. Under typical operating conditions, air is supplied at about 50°C and 1.2 atm absolute pressure.

3.2.1.2 CFB riser

The riser section in the CFB is 0.159 m square in cross-section and 2.44 m tall. Most of the riser is made of 12.5-mm thick clear polycarbonate sheet lined with a 3-mm thick clear acrylic sheet. The acrylic sheet is removable and allows for easy replacement should the particles cause excessive wear on the walls. Clear wall materials were used in order to facilitate visualization of internal hydrodynamic behavior, and the upper half of one of the four walls is removable to allow for flexibility in the experiments (e.g., installation of surfaces for heat transfer measurements, alternative flow visualization schemes).

3.2.1.3 Particle separation

As the gas-solid mixture exits the riser, a separation system is required to remove the solid particles for recirculation. In this apparatus, the separation occurs in two stages. The first separator is an inertial separator, where an array of beams partially obstructs the flow. As particles hit these beams, they drop into the downcomer; this type of separator was also used in the 2-MW combustor. This separator is not completely effective, so a centrifugal separator is installed downstream of the inertial separator. A pneumatic transport line carries particles collected from the centrifugal separator to the downcomer.

3.2.1.4 Particle collection and recirculation

Solid particles separated from the flow exiting the riser are collected in a standpipe, also called the downcomer. In this CFB, the downcomer is a 10-cm diameter pipe made of clear acrylic. The translucence allows for the observation and measurement of the solid feed rate into the riser. The downcomer bends at a right angle in order to feed particles into the riser; this feed system is known
as an "L-valve" because of its shape. To prevent plugging and to facilitate control of the feed rate, the L-valve is slightly aerated with pressurized air.

3.2.1.5 Air filtration and exhaust
Some of the finer particles may still be entrained in the air flow as it exits the centrifugal separator, resulting in the need for a filtration system. The air exiting the separators is directed into a box, in which the cross-sectional area of the box is large in order to reduce the gas velocity and improve the effectiveness of the filtration system. The box contains two filters for redundancy, and the air exhausts to ambient.

3.2.2 Particle characteristics
In order to properly describe the attributes of a fluidized bed, it is necessary to quantify certain properties of the particles. These include the average particle size and the statistical distribution of sizes, the sphericity of the particles, the density of the solid material and the minimum fluidization velocity. Three types of particles were used in this project: steel powder with an average size of 69 μm, from Atlantic Engineering Co. of Bergenfield, NJ; sand with an average size of 128 μm, from Ottawa Sand Co. of Ottawa, IL; and sand with an average size of 182 μm, also from Ottawa Sand. The paragraphs below describe methods for measuring the aforementioned parameters, and Appendix A contains a summary of the results.

3.2.2.1 Particle size and distribution
The size of the particles can be determined using a sieving system. This consists of stackable trays that have screened bottoms; the fineness of the screen mesh varies systematically from tray to tray. By stacking the trays properly, filling the top one with a sample of particles and shaking the stack for some period of time, the particles will distribute themselves among the trays. Although exact particle sizes cannot be determined this way, the particle sizes can be closely estimated. For example, if the screen on one tray has openings that are 53 μm and the screen on another tray has openings that are 44 μm, then the particles that fell through the coarser screen but remain on the finer screen must be between 44 and 53 μm diameter. By assigning the average of the mesh sizes to the mass of particles remaining, a statistical distribution of particle sizes can be determined. Furthermore, the mean particle size, as typically calculated for fluidized beds, corresponds to the average surface-to-volume ratio, as follows [Grace, 1982]:

\[
\frac{1}{d_p} = \sum \frac{x_i}{d_i}
\]

(3.1)
where \( d_p \) is the mean particle size, \( x_i \) is the mass fraction of the \( i \)th sample, and \( d_i \) is the diameter corresponding to the \( i \)th sample. Table A.1 and Figures A.1-3 contain the mean particle sizes and distributions for the solid particles used in this study.

3.2.2.2 Particle sphericity

The sphericity of the particles characterizes their roundness and is a parameter important for geometric similarity in fluidized beds. Sphericity is defined as the ratio between the surface area of a sphere and the surface area of a particle having the same volume as the sphere [Kunii & Levenspiel, 1969]. Sphericities for particles in this study were measured following a similar approach as Farrell [1996], in which the circularity (the equivalent of sphericity in two dimensions) of the particles is measured via microscopic investigation of photographs of many particles. Details concerning the actual measurements are given by Devendran [1997], and the results are presented in Table A.1 in Appendix A. The average sphericities of the particles in this study are between 0.7-0.8, which is typical of particles in fluidized beds.

3.2.2.3 Solid density

The density of the solid particles was measured using a helium pycnometer. This device uses ideal gas principles to calculate the volume occupied by a sample of solid particles; the inert gas helium is used to avoid any contamination of the sample. A chamber is filled with a known mass of particles and then installed in the pycnometer. A chamber of equal empty volume is filled with helium at some pressure, and then that amount of helium is transferred to the sample chamber, where a higher pressure is required to fill the reduced volume. Given that the system is isothermal, the difference in pressure between the two chambers directly indicates the volume occupied by particles and therefore the density of the known mass of particles. The results of the pycnometry measurements are given in Table A.1 of Appendix A. For the steel particles, \( \rho_s = 6980 \text{ kg/m}^3 \) and for the sand particles, \( \rho_s = 2650 \text{ kg/m}^3 \).

3.2.2.4 Minimum fluidization velocity

Due to inaccuracies in the flowmeter in one of the test facilities, measurements for the minimum fluidization velocity were unobtainable. Typically, one simply measures the variation in pressure drop with gas superficial velocity across a bed of particles; the point at which the pressure drop becomes constant is the point of minimum fluidization. The constant pressure drop indicates an equality between the momentum loss of the gas and the weight of the particles. The following correlation is given by Grace [1982] to calculate the minimum fluidization velocity based on known properties of the gas and solid particles:
\[
\frac{\rho f \mu_{nf} d_p}{\mu} = \sqrt{740 + 0.0408 \frac{\rho_s \rho f g d_p^3}{\mu^2}} - 27.2
\]  
(3.2)

This correlation is based on an equation developed by Ergun [1952], with simplifications made based on empirical observations of fluid-particle systems. Table A.1 in Appendix A contains the values of \( u_{nf} \) for the solid particles used in this study.

3.2.3 Basic diagnostic equipment

The CFB apparatus also contains some standard instrumentation to measure various operating parameters, including the gas superficial velocity \( (u_0) \), the solid recycle rate \( (G_s) \), and the average volumetric solid concentration.

3.2.3.1 Gas superficial velocity

The gas superficial velocity is defined as the volumetric flow rate divided by the cross-sectional area of the riser. The volumetric flow rate is measured in the air supply line using an orifice plate to create a pressure drop that can be related to the flow rate [see Beckwith, et al., 1993, e.g.]. In this case, the pipe has a 10 cm inner diameter and the orifice is concentric with a diameter of 5.62 cm. The pressure differential is measured with a water manometer from locations four inches (one pipe diameter) upstream of the orifice to two inches (one-half a pipe diameter) downstream. The initial pressure and temperature of the air are measured using gauges installed in the piping, and these properties are used to determine the density and viscosity of the air. The flow rate, superficial velocity, density and viscosity of the air were calculated using the measurements of temperature, pressure and pressure drop, following procedures developed by Miller [1989] and Farrell [1996].

3.2.3.2 Solid recirculation rate

The solid recirculation rate is measured by observing the rate at which particles flow through the downcomer; this technique was used previously by Westphalen [1993]. This technique is based on the assumption that the particles descend in plug flow, or at uniform velocity across the downcomer, and it requires knowledge of the loosely-packed void fraction of the moving particles. The latter can be measured by pouring particles into a graduated cylinder, weighing the sample, and measuring the volume occupied. If \( u_{dc} \) is the velocity of the particles measured in the downcomer, the solid recycle rate expressed as mass flow per unit area is given by:

\[
G_s = \frac{\rho_s (1 - \varepsilon_p) u_{dc} A_{dc}}{A_{riser}}
\]  
(3.3)
where \( \varepsilon_{lp} \) is the loose-packed void fraction of the particles. For the particles used in this study, measurements of \( \varepsilon_{lp} \) are given in Table A.1 in Appendix A.

### 3.2.3.3 Volumetric solid concentration

One parameter that is typically of interest to CFB designers is the concentration of solid particles in the riser and the variations in concentration along the length of the riser. The concentration of particles in a given section of the riser can be calculated by measuring the drop in air pressure along that section. In a steady state, any loss in gas momentum will manifest itself as a drop in pressure. The main contributions to the pressure loss include gas mixing at the entrance of the riser, shear stresses at the walls and the weight of the solid material. In this CFB, the gas enters the riser uniformly distributed about the cross-section, so that mixing effects are small. Furthermore, wall shear forces are small when compared to the weight of typical CFB particles, even at concentrations as low as 0.1% by volume. Therefore, the measured pressure drop can be equated to the average solid concentration as follows:

\[
\Delta P = \rho_s (1 - \varepsilon_{avg}) g \Delta x
\]

(3.4)

where \( (1 - \varepsilon_{avg}) \) is the volumetric solid concentration averaged over the cross-section of the riser and \( \Delta x \) is the distance over which the pressure drop is measured. Solving for \( (1 - \varepsilon_{avg}) \) in Eq. (3.4) yields:

\[
(1 - \varepsilon_{avg}) = \frac{\Delta P}{\rho_s g \Delta x}
\]

(3.5)

The CFB in this study has eleven pressure taps along the length of the riser; the openings are slanted upward at an angle of about 45° to prevent the downward-moving particles near the wall from clogging the taps. Eleven taps allow for ten differential pressure measurements which were made using pressure transducers from Autotran, Inc. of Hopkins, MN in conjunction with data acquisition hardware and software from Keithley Instruments, Inc. of Taunton, MA. Table B.1 in Appendix B contains more detailed information concerning the location of the pressure taps and the calibration of the pressure transducers. When measuring the solid fraction in the riser, pressure readings were taken at a rate of 100 Hz for 15 seconds and then averaged over that span for use in Eq. (3.5).

### 3.2.4 Summary of CFB design specifications

The preceding paragraphs have described in some detail the essential elements in the design and construction of a scale-model CFB. Among these are the major sub-systems required to make the apparatus functional as well as the basic diagnostic equipment to measure operating conditions.
Knowledge of the latter parameters is necessary for any experimental set-up, especially one that is specifically designed as a scale model. As seen in the previous chapter, certain dimensionless operating parameters govern hydrodynamic behavior in a CFB, and knowing the values of these dimensionless groups is the key to proper application of any experimental results.
3.3 DESCRIPTION OF EXPERIMENTAL METHODS

3.3.1 Overview – thermal image velocimetry (TIV)
In order to fulfill the goals of this project, a technique to observe the flow at the wall of a CFB needed to be developed. Given the concentration of particles in the system, conventional video methods failed to provide the level of contrast required. Hori and Kuroki [1994] used a laser sheet with conventional video to visualize flow in a CFB; however, the concentration level was extremely dilute, with values of \( G_z \) that were roughly 100 times lower than that typically found in CFBs. Even under dilute conditions, it can be impossible to fix an origin of the motion of individual clusters without some sort of tracer mechanism. A technique called thermal image velocimetry (TIV) was developed to overcome these obstacles.

The basic concept behind a TIV system is that heating the clusters at the wall alters their emission of thermal radiation and thereby distinguishes them from the rest of the bed. With the appropriate detection device or devices, properties of the flowing clusters can be measured. TIV has several advantages, as follows. It is a non-invasive method for marking the clusters and tracking their motion at the wall, as TIV does not require any instrumentation in the flow path. Also, TIV distinguishes the clusters at the wall in the heated section from clusters coming to the wall below the heater. Further, it does not require the addition of tracer particles to the flow to enhance flow visualization, which is beneficial since such particles might affect the hydrodynamics in the riser. Finally, the simple method of marking the clusters with heat does not require the bed material to be any special material itself (such as a photo-luminescent substance). The latter attribute allows for the selection of a material which will yield similitude with any desired full-sized configuration.

3.3.2 Conceptual design of a TIV system
In general, a TIV system in a CFB requires a test section containing three items: a heater for the clusters; a system capable of detecting infrared radiation; and a wall that is transparent to infrared radiation. Figure 3.2 shows a general schematic of a TIV test section. The heater must be flush with the wall of the CFB so that the flow of the solids and the gas remains undisturbed, since small roughness elements on the wall have been shown to disrupt the flow of clusters at the wall [Lints, 1992]. The heater will also need to have enough power to raise the temperature of the clusters such that they do not cool down to the ambient bed temperature before they reach the end of the test section. As a cluster travels at the wall after heating, its radiant emission is detectable. When the cluster moves away from the wall its radiant signal will be obstructed by the new material at the wall and it will no longer be detected; however, its motion is no longer of interest since it is no longer contributing to heat transfer at the wall. The TIV technique allows for the viewing of the
history of the clusters which have passed the heater, and it distinguishes them from those clusters that arrive at the wall below the heater.

When a cluster is heated at the wall, its temperature is raised tens of degrees above the ambient level in the bed. When running a cold scale-model CFB, these temperatures are on the order of 300 K, so that most of the thermal radiation emitted is in wavelengths around 10 μm. Because of this, special infrared detectors or infrared cameras are required to observe the flow of the marked clusters. Also, most types of glass and plastic materials available for wall materials absorb all energy at wavelengths above roughly 2 μm. Therefore, a special wall is required to allow for the transmission of the radiant signal from the solid material at the wall. The specific design of the TTV test section for these experiments is discussed next.

3.3.3 Construction and implementation of the TIV system
3.3.3.1 Test section location
In these experiments, the TTV test section is located in the upper portion of the CFB, from 0.92 m above the distributor to the top of the riser (roughly the upper 60% of the bed). The TIV test section is a plane wall, simply replacing the removable wall in that portion of the square bed. The top of the heating section is about 0.48 m from the top of the bed, and it extends 0.15 m down from there. The lower 0.89 m of the test section (as well as the upper 0.48 m) contains the infrared-transparent wall for viewing the motion of the clusters. For the most part, the viewing is done below the heated plate, since the clusters move downward. However, the section above the heated plate is also transparent to infrared radiation so that any upward motion of clusters can be
observed (the elevation of the infrared camera can be varied). Figure 3.3 shows a schematic of this TTV test section.

![Diagram of TTV test section](image)

**FIGURE 3.3: Schematic for the TTV test section in this CFB**

### 3.3.3.2 Heating element
The heating section consists of a resistance heater attached to a thin aluminum plate, where the aluminum plate is the wall of the CFB in that section. The aluminum plate is 0.15 m square and 0.5 mm thick, and the power output of the heater is 190 W. It can be calculated that this level of power imparts roughly a 20 K temperature rise to the surfaces of the clusters that contact the plate (this analysis is discussed later).

### 3.3.3.3 Viewing sections
The viewing sections are made of low-density polyethylene (LDPE), also 0.15 m wide and 0.5 mm thick; the low-density polyethylene sheets were bought from Laird Plastics of Woburn, MA. LDPE was selected as the wall material for its relatively high transmissivity to long-wavelength infrared radiation. Whereas typical glass, acrylic, or polycarbonate plastics absorb all radiation at
wavelengths above 2 μm, the 0.5 mm-thick LDPE is roughly 40-50% transparent to radiation at wavelengths of about 10 μm, which is the region of interest for room-temperature experiments. Since measuring the temperature of the clusters is not the primary concern, the level of signal attenuation can be ignored as long as that signal is still strong enough to allow for the detection of clusters. With proper compensation for attenuation, however, the temperature of the clusters can be measured; these measurements are useful for validating the measurement of contact times (discussed later).

A smooth transition between the heating and viewing sections is ensured by making the aluminum plate for the heater and the LDPE wall the same thickness (0.5 mm). Any discontinuity where the two materials meet is filled in with a putty which was sanded down to the same level as the aluminum and LDPE surfaces. A grid-shaped piece of plastic supports the LDPE sheets, where the grid size of 1 cm x 1 cm allows for nearly unobstructed viewing. The LDPE sheet is continuous, but there are a discrete number of viewing regions, dictated by the location of the brackets that hold the wall to the rest of the riser. Figure 3.4 shows the details, from the front and the side, of the construction of the test section. With some applied tension in the LDPE sheet, the grid minimizes the distortion that results from the slight pressurization in the bed. It is unlikely that distortion is eliminated completely; in fact, it can be estimated that the maximum deflection of the LDPE in the grid is on the order of tens of microns [using formulae from Roark & Young, 1975]. These perturbations on the wall surface are one possible source of error in these experiments, but since the perturbations are continuous and not abrupt they should cause less disruption of the flow. Experimental observations confirm this.
viewing windows:
8 windows,
= 0.11 m square,
with 1 cm x 1cm grid
for reinforcement

brackets to hold
wall to riser (10 total)

heating
section

LDPE sheet,
0.5 mm thick

aluminum plate,
0.5 mm thick,
15 cm square

poly carbonate frame
(12.5 mm thick)
with cut outs for
viewing windows

front view
(from outside)
side view

(inside of riser)

FIGURE 3.4: Construction details for the wall in the TIV test section (not to scale)
3.3.3.4 Infrared detection system

The infrared detection system consists of an infrared camera and a VCR to record the signal. The camera is the Model 600 Infrared Imaging Radiometer from Inframetrics, Inc. of Billerica, MA, which has a mercury/cadmium/telluride (HgCdTe) detector that can be set to detect radiation in the 8-12 μm portion of the electromagnetic spectrum. The output is a video signal at the standard video rate of 30 Hz; this is wired into a conventional VCR. Since the camera is mounted on a tripod, it can be aligned with any of the viewing windows and it can be set back at such a distance to capture the entire window in its field of view. With conventional video output, the data is reduced by analyzing the videotape in slow motion., and Figure 3.5 shows sample pictures of clusters from the infrared camera; these are still frames captured from the video output.

![Infrared images of heated clusters](image)

**FIGURE 3.5:** Infrared images of heated clusters

With the infrared camera, higher temperatures show up whiter in color, so that the "white" objects in Fig. 3.5 are actually the heated clusters. The "black" grid that appears is the grid supporting the LDPE, both of which are at ambient temperature.
3.4 DATA ACQUISITION AND REDUCTION

The TIV technique can be used to measure both the velocities of clusters \( u_d \) and the residence time of clusters at the wall \( t \). The following sections describe the methods for measuring these parameters.

3.4.1 Measuring cluster velocities

The cluster velocities are measured by keeping the camera focused on a particular viewing port below the heater for some period of time. The camera samples images at 30 Hz, so that the number of video frames required for a cluster to pass in and out of a viewing port indicates the time. Each viewing port has a known length, and dividing by the time for a cluster to pass through the field of view yields the velocity in that region. These velocities can be averaged for a number of clusters passing through that viewing port in the sampling time. Furthermore, there are five viewing ports below the heating section (shown in Fig. 3.4), allowing for the measurement of any change in cluster velocities as they move along the wall. In a more sophisticated TIV system, the infrared camera could be replaced by two infrared detectors, allowing for cross-correlation techniques to be utilized in the measurement of cluster velocities.

The measured velocities are based on an average of about ten or twelve clusters observed in each window for each operating condition. It should be noted that there is a fairly large error in the velocity measurements, roughly \( \pm 30\% \) for each measurement of average velocity. About 10% of that results from confidence limits on the statistical distribution of the data, while the bulk of the error results from the relatively low sampling rate of the camera. Since the viewing ports are about 0.11 m long and the clusters move at about 1 m/s, it only takes about three frames for a cluster to pass through the field of view. The absolute error can be estimated at one-half of a frame, so that the precision error is about 20%.

3.4.2 Measuring cluster-wall contact times

Similar to the measurement of cluster velocities, cluster-wall contact times are measured by focusing the infrared camera on each viewing port (see again Fig. 3.4) for a certain period of time. By analyzing the resulting videotape in slow motion, the average number of clusters passing by a particular viewing port in a given period of time can be measured; this is called the "passing frequency" of clusters (a cluster is defined by the leading edge observed to pass along the viewing port, and there is obviously some subjectivity in this definition). In analyzing the videotape, the top and bottom halves of the first viewing port are analyzed separately, allowing for slightly greater resolution of the passing frequencies. To retain consistency with the nomenclature thus far, the
two halves of Viewing Port A can be referred to as A1 and A2 for the top and bottom halves, respectively.

Data for passing frequencies are obtained by sampling 60 seconds of video data at 30 Hz in each viewing port. In analyzing the videotape, each 60-second sample is subdivided into about 10 sets; the average passing frequency for each of these 5-6 second sets was measured, and then that group is averaged to yield an overall average. Subdividing the 60-second sample into a number of sets allows for the estimation of statistical confidence limits on the group average; this was typically found to be about ±10% for 90% confidence limits. Furthermore, the visual analysis of a sample was determined to be repeatable within about 5%, so that the total uncertainty on a given average passing frequency is about ±15%. By using the velocity history of the clusters to convert from distance to time, the uncertainty error associated with the velocity measurements is introduced (roughly ±30%). Using a root-mean-square method for combining uncertainties [Beckwith, et al., 1993], the total uncertainty in the measurement of the contact times is ±35%.

3.4.2.1 Contact time below the heater
The viewing ports are located at fixed distances below the heated plate; therefore, the passing frequency in each viewing port actually represents the passing frequency as a function of the distance traveled after heating. Since the velocity history of the clusters below the heated plate is known for each operating condition (as discussed previously), the passing frequency is also known as a function of the travel time below the heated plate. If normalized, the passing frequency can be thought of as a probability function for observing a cluster a certain time after heating. With that, the rate of change of the passing frequency, or the "shedding function", is the probability function for a cluster disappearing from the wall a certain time after heating. Restated, this is the probability that a cluster remains in contact with the wall a certain time after heating, which is precisely the information being sought.

Figure 3.6 shows the passing frequency function for one operating condition, the shape of which is typical of all of the operating conditions. The type of curve that fits the passing frequencies best is an exponential function; normal and uniform distributions were also considered [Panta, 1996]. Since the shedding function is the negative of the derivative of the passing frequency, the shedding function will also have an exponential shape with the same time constant. If the shedding function is normalized such that the area under its curve is unity, then the shedding function is equivalent to the probability that a cluster travels a certain time after heating. This normalized shedding function is also shown with the measurements in Fig. 3.6.
An exponential probability function, like the normalized shedding function that has been calculated, is defined by:

$$p_s(\tau) = \frac{1}{\tau_{avg}} \exp\left(-\frac{\tau}{\tau_{avg}}\right)$$ \hspace{1cm} (3.6)

where $p_s$ is the probability density function for cluster shedding, $\tau$ is the random variable and the average value is simply $\tau_{avg}$; this is the average contact time of a cluster at the wall below the heater. A probabilistic analysis (presented next) shows that the average total contact time at the wall is simply twice the average contact time after heating.

### 3.4.2.2 Extrapolation of results to total contact time

Since the heated clusters could begin their travel down the wall at or above the heater, the calculation of the total contact time must account for this. In order to calculate the total contact time from the measured contact time, a probabilistic analysis is applied. For this, two assumptions are made: one is that there is some arbitrary distribution of cluster contact times; the other is that the clusters may have passed the bottom of the heating section at any portion of their travel. Restating the second assumption, there is a uniform probability that a particular cluster will have passed the bottom of the heating section at the beginning, the middle, or the end of its life at the wall, or any fraction thereof in between. Both of these assumptions have been made before for flow near the
The wall of CFBs [see Fang, et al., 1995a,b]. The first one can be justified by the fact that some distribution is seen in the partial or measured contact times as in Fig. 3.6. The second assumption is based on the concept of uniform deposition of clusters along the wall; the justification for this assumption is that we would expect deposition rates to be a function of the local cross-sectional concentrations, which are roughly constant in the test section (the upper portion of the riser).

The probabilistic analysis starts with the assumption that the total contact times of clusters at the wall follow some probability density function for the distribution of total contact times \( p_c(t) \), with some maximum contact time \( t_{max} \), such that:

\[
\int_0^{t_{max}} p_c(t) \, dt = 1
\]  \( \text{(3.7)} \)

There will be a different distribution of measured contact times, \( p_d(\tau) \), where \( p_s \) and \( \tau \) represent the probability density function for the distribution of measured contact times as in Eq. (3.6). As shown in Appendix C, \( p_d(t) \) and \( p_d(\tau) \) are related by:

\[
p_s(\tau) = \int_{\tau}^{t_{max}} \frac{p_c(t)}{t} \, dt
\]  \( \text{(3.8)} \)

Differentiating Eq. (3.8) yields the functional form of \( p_c(t) \) from the functional form of \( p_d(\tau) \):

\[
p_c(t) = -t \frac{dp_s}{d\tau}
\]  \( \text{(3.9)} \)

Since the functional form of \( p_d(\tau) \) has been determined from Fig. 3.6 and Eq. (3.6), the general form of \( p_c(t) \) is found to be:

\[
p_c(t) = \frac{\tau}{\tau_{avg}^2} \exp \left( -\frac{t}{\tau_{avg}} \right)
\]  \( \text{(3.10)} \)

This functional form of \( p_c(t) \) is that of a gamma distribution with a parameter \( n = 2 \) [Lapin, 1983]; Figure 3.7 shows a plot of a typical gamma distribution. The fact that cluster residence times at the wall might follow a gamma distribution was first proposed by Fang, et al. [1995a]. Until now, however, there had been no evidence to prove this to be the case. Analysis by Panta [1996] shows that assuming a gamma distribution for \( p_c(t) \) clearly yields the best results for \( p_d(\tau) \). Panta used the chi-squared goodness-of-fit test [Lapin, 1983] to compare the measured data shown in Fig. 3.6 to the function \( p_d(\tau) \) generated using a function \( p_c(t) \) that is either a gamma, exponential, uniform or normal distribution. The value of the test statistic for the different assumed distributions were:
gamma, 0.027; exponential, 0.088; uniform, 0.24; and normal, 1.69. The lower the value, the better the confidence in the fit. Since the gamma distribution can be derived analytically and has the best fit empirically, it can be confidently concluded that the gamma distribution represents the distribution of actual cluster-wall contact times.

![Graph of gamma distribution](image)

**FIGURE 3.7:** Typical gamma distribution with $n = 2$ and $t_{avg} = 2$ s

The average total contact time can be determined from the results of the preceding analysis. Starting with the definition of $t_{avg}$:

$$t_{avg} = \int_{0}^{t_{max}} \tau \ p_s(\tau) \ d\tau \quad (3.11)$$

Substituting Eq. (3.9) into Eq. (3.11) and integrating by parts yields $t_{avg} = 2t_{avg}$ for any distribution $p_c(t)$ (see Appendix C). Obviously, this result can be shown to be true specifically for the gamma distribution of Eq. (3.10) as well.

3.4.2.3 Correction for cooling of clusters

When a cluster disappears from view, or when a change in passing frequency is measured, it has been assumed that the cluster was shed from the wall. Since the detection of clusters depends on their temperature, it may be possible that the disappearance of clusters results from cooling. Additional measurements and analysis show that the heater has enough power to sufficiently raise
the temperature of almost all of the clusters such that they can be observed, or such that they have not cooled down, by the end of the entire test section. The proportion of clusters that are not heated sufficiently is shown to be negligible. This ensures that the measured changes in the passing frequency are due to clusters moving away from the wall, and not from clusters cooling down. In reality, the clusters that begin their contact with the wall near the end of the heating section are more likely to be insufficiently heated. In this section, an analysis and additional results from the scale-model CFB show that the error introduced by insufficient heating is very small.

The temperature rise that the surface of a heated cluster must maintain over the ambient bed temperature in order to be seen at the bottom of the test section can be estimated from the characteristics of the infrared camera. The camera output is 16 shades of gray over a 5°C range, but since the data reduction method is visual rather than digital, a resolution of 2.5°C (3 shades – dark, gray, white) is assumed for conservatism. Therefore, the problem can be recast in three parts: how much heating time is required so that the cluster surface has a 2.5°C temperature rise at the end of the test section; how many clusters are heated less than this time; and how many of these clusters actually stayed at the wall after they cooled below the 2.5°C threshold. The latter two parts require knowledge of the probability function for cluster travel which was revealed in the previous section.

The first part of the problem is to determine how much time the clusters need to be heated to still be detected at the end of the test section. To do this, the one-dimensional unsteady energy equation can be solved for clusters that are homogeneous bodies with a given set of physical properties, as they are assumed to be in the particle-convective heat transfer model. Neglecting radiant heat transfer since the temperatures are relatively low, the governing equation is:

$$\frac{1}{\alpha_{\text{cluster}}} \frac{\partial T}{\partial t} = \frac{\partial^2 T}{\partial x^2}$$

(3.12)

Assumptions in this model are that: the heated plate is roughly at a uniform temperature determined by the power input, yielding $T_{\text{plate}} - T_{\text{bed}} = 70°C$; the contact gap between the cluster and the plate is the size of a couple of particles, yielding $h_{\text{contact}} = 200 \text{ W/m}^2\text{-K}$; the controlling heat-transfer mechanism below the heating section is natural convection on the outside of the LDPE wall, yielding $h_{\text{free conv}} = 4 \text{ W/m}^2\text{-K}$; the cluster contains about 7% solid particles by volume [Lints & Glicksman, 1993] yielding $\alpha_{\text{cluster}} = 10^{-7} \text{ m}^2/\text{s}$; and the cluster is about 1 mm thick. For the assumption regarding natural convection on the outside of the LDPE, it can be shown that the properties of LDPE are such that the thermal response to a given cluster is nearly an order of
magnitude slower than the passage time for a cluster, so that a steady-state condition exists for the natural convection. These assumptions yield the following boundary conditions:

\begin{align}
\text{at } t = 0, \quad & T_{\text{cluster}} = T_{\text{bed}} \text{ or } \Delta T_{\text{cluster}} = 0 \\
\text{for } t \leq t' \text{ and } x = 0, \quad & q = h_{\text{contact}} (T_{\text{plate}} - T_{x=0}) \\
\text{for } t > t' \text{ and } x = 0, \quad & q = h_{\text{free conv}} (T_{\text{room}} - T_{x=0})
\end{align}

where \( t = 0 \) represents the time at which a cluster initiates contact with the heated wall and \( t' \) is the time at which a cluster passes the bottom of the heating section.

To arrive at a solution for Eq. (3.12), a forward-difference technique was used in conjunction with a spreadsheet program. If the cluster travels the full length of the heating section, then the solution for Eq. (3.12) can be compared to measurements of the maximum temperature of the clusters as seen below the heating section; these results are shown in Figure 3.8.

![Graph showing maximum cluster surface temperatures, measured vs. predicted](image)

**FIGURE 3.8:** Maximum cluster surface temperatures, measured vs. predicted

The temperature measurements in Fig. 3.8 were obtained after compensating the output of the infrared camera for the emissivity, transmissivity, and reflectivity of the LDPE wall; the compensation analysis can be found in Appendix D. The measurements of cluster temperatures were obtained under typical operating conditions. Based on this comparison, it appears that the forward-difference model is sufficiently accurate to estimate the minimum heating time that clusters
require (i.e. the heating time that yields $\Delta T_{\text{cluster}} = 2.5^\circ\text{C}$ at the end of the test section). This time is $t_{\text{min}} = 0.05$ s, which is small compared to roughly 0.15 s for full contact with the heater.

Based on this result, the probabilistic model for cluster-wall contact can be used to determine how much error was introduced by the fact that some of the clusters were insufficiently heated. In knowing $p_e(t)$ from Eq. (3.10) and Fig. 3.6, one can calculate the percentage of clusters in each viewing port that started their descent below the minimum heating point and disappeared from view as a result of cooling instead of moving away from the wall. This analysis was carried out by Panta [1996], and the results can be used to correct the passing frequency function as shown in Figure 3.9.

![Figure 3.9: Cluster passing frequency function, corrected for cluster cooling](image)

This correction is small, and it changes the decay constant – which is equal to the average contact time below the heating section – by 0.5% to 3.0% [Panta, 1996]. Since the error is small and is also well within the uncertainty limits of the experiments, it is considered negligible. As a result, the total average cluster-wall contact time can be obtained simply by doubling the contact time measured below the heating section as determined earlier.
3.4.3 **Summary of data reduction methods**

The methods for measuring cluster velocities and contact times are fairly simple, even though there is complexity in justifying some of the underlying assumptions. For velocities, TIV is used in a rather straightforward manner, measuring the time required for a cluster to travel a certain distance. For contact times, TIV is used to measure the variation in frequency at which clusters pass along the test section. The decay constant for the variation in frequency is equal to one-half the average cluster-wall contact time.
3.5 REFERENCES


S. Ergun, Fluid flow through packed columns, Chemical Engineering Progress, 48, 89 (1952).


4. EXPERIMENTS: RESULTS AND DISCUSSION

4.1. EXPERIMENTAL OPERATING CONDITIONS

Measurements of cluster velocities and cluster-wall contact times were made under a variety of operating conditions, in order to better understand the physical mechanisms governing each of the phenomena. Table 4.1 presents a matrix of operating conditions and dimensionless operating conditions along with the measured average solid concentration in the TIV test section, where \( u* = u_0/u_mf \) and \( G* = G_y/\rho_s u_0 \) are the dimensionless gas velocity and solid recycle rate.

<table>
<thead>
<tr>
<th>Case number</th>
<th>Bed material</th>
<th>( u_0 ) (m/s)</th>
<th>( G_y ) (kg/m(^2)s)</th>
<th>( u* )</th>
<th>( G* )</th>
<th>avg. solid conc.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>69 ( \mu )m steel</td>
<td>2.3</td>
<td>9.3</td>
<td>190</td>
<td>5.8 \times 10^{-4}</td>
<td>0.25%</td>
</tr>
<tr>
<td>2</td>
<td>&quot;</td>
<td>2.8</td>
<td>11</td>
<td>230</td>
<td>5.6 \times 10^{-4}</td>
<td>0.21%</td>
</tr>
<tr>
<td>3</td>
<td>&quot;</td>
<td>2.8</td>
<td>19</td>
<td>230</td>
<td>9.7 \times 10^{-4}</td>
<td>0.46%</td>
</tr>
<tr>
<td>4</td>
<td>&quot;</td>
<td>2.7</td>
<td>30</td>
<td>220</td>
<td>1.6 \times 10^{-3}</td>
<td>0.56%</td>
</tr>
<tr>
<td>5</td>
<td>&quot;</td>
<td>3.3</td>
<td>18</td>
<td>270</td>
<td>7.8 \times 10^{-4}</td>
<td>0.24%</td>
</tr>
<tr>
<td>6</td>
<td>&quot;</td>
<td>3.3</td>
<td>28</td>
<td>270</td>
<td>1.2 \times 10^{-3}</td>
<td>0.42%</td>
</tr>
<tr>
<td>7</td>
<td>&quot;</td>
<td>3.6</td>
<td>25</td>
<td>300</td>
<td>9.9 \times 10^{-4}</td>
<td>0.28%</td>
</tr>
<tr>
<td>8</td>
<td>&quot;</td>
<td>3.6</td>
<td>34</td>
<td>300</td>
<td>1.4 \times 10^{-3}</td>
<td>0.49%</td>
</tr>
<tr>
<td>9</td>
<td>128 ( \mu )m sand</td>
<td>2.5</td>
<td>13</td>
<td>160</td>
<td>2.0 \times 10^{-3}</td>
<td>0.63%</td>
</tr>
<tr>
<td>10</td>
<td>&quot;</td>
<td>2.5</td>
<td>21</td>
<td>160</td>
<td>3.0 \times 10^{-3}</td>
<td>0.89%</td>
</tr>
<tr>
<td>11</td>
<td>&quot;</td>
<td>2.9</td>
<td>11</td>
<td>180</td>
<td>1.5 \times 10^{-3}</td>
<td>0.51%</td>
</tr>
<tr>
<td>12</td>
<td>&quot;</td>
<td>3.2</td>
<td>14</td>
<td>200</td>
<td>1.7 \times 10^{-3}</td>
<td>0.46%</td>
</tr>
<tr>
<td>13</td>
<td>&quot;</td>
<td>3.3</td>
<td>18</td>
<td>210</td>
<td>2.1 \times 10^{-3}</td>
<td>0.54%</td>
</tr>
<tr>
<td>14</td>
<td>&quot;</td>
<td>1.8</td>
<td>11</td>
<td>110</td>
<td>2.4 \times 10^{-3}</td>
<td>0.90%</td>
</tr>
<tr>
<td>15</td>
<td>182 ( \mu )m sand</td>
<td>2.9</td>
<td>15</td>
<td>90</td>
<td>1.9 \times 10^{-3}</td>
<td>0.75%</td>
</tr>
<tr>
<td>16</td>
<td>&quot;</td>
<td>3.4</td>
<td>15</td>
<td>100</td>
<td>1.6 \times 10^{-3}</td>
<td>0.61%</td>
</tr>
</tbody>
</table>

TABLE 4.1: Experimental operating conditions

The gas superficial velocity and solid recycle rate were varied to provide a range of conditions for both, while holding as many other parameters constant as possible. In addition, some attempt was made to match dimensionless parameters – \( u* \) and \( G* \) – between cases.
4.2 MEASUREMENTS OF CLUSTER VELOCITIES

4.2.1 Results

Figure 4.1 shows the variation of cluster velocity with time at the wall below the heating section, where the velocity history is used to convert distance along the wall to time at the wall.

![Graph showing cluster velocity history](image)

**FIGURE 4.1:** Average cluster velocity history in TIV test section

The data presented in Fig. 4.1 are for the first eight cases, all with steel particles, and each symbol on the graph represents a different operating condition. The terminal velocity for the steel particles is roughly 1.2 m/s. Similar velocities and trends were observed with the sand particles, with terminal velocities of roughly 1.2 m/s for the 128 µm sand and 1.1 m/s for the 182 µm sand. As noted earlier, there is a fairly large amount of error in the velocity measurements – roughly ±30% for each measurement of average velocity.
4.2.2 Discussion
4.2.2.1 Direction of cluster motion
Although it is not explicitly shown in any of the tables or figures, the direction of the motion of clusters was observed to be downward. Some video segments were taken in the viewing port immediately above the heating section in order to investigate the possibility of upward motion, and no clusters were observed in these videos. This observation (or lack thereof) was made under three different sets of operating conditions that are representative of the first eight sets listed in Table 4.1: \( u_0 = 3.0 \) m/s and \( G_s = 10 \) kg/m\(^2\)-s, \( u_0 = 2.9 \) m/s and \( G_s = 20 \) kg/m\(^2\)-s, and \( u_0 = 2.6 \) m/s and \( G_s = 10 \) kg/m\(^2\)-s. The downward motion of the clusters is consistent with the core-annular flow behavior in CFBs that has been widely observed.

4.2.2.2 Cluster acceleration
The fact that the velocity of the clusters increases the longer they travel at the wall can be observed in Fig. 4.1. The clusters have been at the wall for some period of time prior to passing the end of the heating section, so that any initial velocity or acceleration cannot be observed. However, acceleration is still observed after heating and this acceleration seems to taper off the longer the clusters are falling, indicating that the observed phenomenon is similar to the free-fall of a body to a terminal velocity in a gravitational field. As discussed earlier, the existence of this phenomenon at the wall of a CFB has been postulated [Glicksman, 1988] and observed with particles descending in an empty tube with upward gas flow [Lints, 1992] but never observed in a fully-operational CFB. Since cluster descent velocities are not constant, it is inaccurate to state that the cluster-wall contact time can be calculated by simply dividing the observed contact length by the measured cluster velocity.

4.2.2.3 Dependence on operating conditions
It is also apparent that the velocities of the clusters are independent of the operating conditions in the CFB. For the eight different sets of gas velocities and solid recycle rates (and resultant solids concentrations), there is very little difference in the measured cluster velocities in any given location along the test section. Whatever variation is observed from case to case certainly falls well within the range of the experimental uncertainty. Furthermore, the phenomenon of acceleration to a terminal velocity is also observed in every case, and this terminal velocity is roughly constant at about 1.2 m/s. This value is comparable to those reported by other investigators, where cluster velocities have been reported between 0.5 and 2.5 m/s, also roughly independent of operating conditions in a given CFB.
4.2.2.4 Free-fall models applied to cluster velocities

Given that the cluster velocities have been observed to be independent of operating conditions and that clusters descend near the wall in a region of low gas momentum, they can be thought of as descending in still air (to a first approximation). With this, two types of drag on a cluster can be considered in order to calculate and predict its velocity history. In one model, the drag force is linearly proportional to the velocity, resulting in the following equation of motion for the cluster:

\[ m_{cl} \frac{du}{dt} = m_{cl} g - C_1 u \]  \hspace{1cm} (4.1)

where \( u \) is the descent velocity of the cluster and \( C_1 \) is the "drag coefficient." Solving Eq. (4.1) for the conditions \( u(0) = 0 \) and \( u(\infty) = u_{cl} \) yields:

\[ u(t) = u_{cl} \left[ 1 - \exp \left( -\frac{g \ t}{u_{cl}} \right) \right] \]  \hspace{1cm} (4.2)

where \( u_{cl} \) is the known average terminal velocity of the clusters. This solution in Eq. (4.2) might be typical when most of the drag on the cluster results from flow through the porous cluster or from a shear interaction with the adjacent wall.

In the other type of drag model, the drag force is proportional to the square of the velocity, as with flow around a solid body at a high Reynolds number. The characteristic size of a cluster is on the order of 1 mm to 1 cm, [see Rhodes, et al., 1992, or Zou, et al., 1994, for example] and a typical descent velocity is between 0.5 and 2.0 m/s [see Rhodes, et al., 1992, or Wang, et al., 1993, for example], so that a characteristic Reynolds numbers might be between 100 and 1000 in a cold-flow scale-model CFB. This is sufficiently high to consider inertial forces in flow over a blunt body and results in the following equation of motion for the cluster:

\[ m_{cl} \frac{du}{dt} = m_{cl} g - C_2 u^2 \]  \hspace{1cm} (4.3)

where \( C_2 \) is the drag coefficient for this case. Eq. (4.3) is difficult to solve analytically, but it can be solved numerically. The solution to this and Eq. (4.2) are superimposed on the measurements of cluster velocities in Fig. 4.1. The time-zero point on Fig. 4.1 is assumed to correspond to \( t = 0.2 \) s for Eqs. (4.2) and (4.3), since passing the heating section nominally represents half of the travel time for a cluster (and the total travel time \( \approx 0.4 \) s for clusters of steel particles, as discussed later). By inspection, it appears that the model for a drag force that is linearly proportional to the velocity characterizes the data best. For comparison, the data of Lints [1992] is shown in Figure 4.2 along with the velocity history predicted by Eq. (4.2), where the origin for cluster travel in Lints' experiments is known.

72
FIGURE 4.2: Linear drag model applied to data of Lints [1992]
4.3 MEASUREMENTS OF CLUSTER-WALL CONTACT TIMES

4.3.1 Results

Figures 4.3, 4.4, 4.5 and 4.6 show the variation of average cluster-wall contact times with solid-gas density ratio, gas superficial velocity, solid recycle rate and average local solid concentration, respectively, under all of the operating conditions listed in Table 4.1. The results in these figures represent the total cluster-wall contact time, as determined from the data reduction methods described in Chapter 3. These figures show that the measured contact times vary between about 0.2 and 0.5 s, with the lower values corresponding to the CFB operating with lighter particles. As noted earlier, there is a fairly large amount of error in the contact time measurements – roughly ±35%. These values for contact times can be used in conjunction with Eq. (2.8) to calculate a convective heat transfer coefficient. Using typical values for the other hydrodynamic parameters in the heat transfer model, one can calculate $h_{conv} = 100 \text{ W/m}^2\text{-K}$, which is consistent with bed-to-wall heat transfer results reported by other researchers.

![Graph showing cluster-wall contact time vs. solid-gas density ratio](image)

**FIGURE 4.3:** Average cluster-wall contact time vs. solid-gas density ratio
FIGURE 4.4: Average cluster-wall contact time vs. gas superficial velocity

FIGURE 4.5: Average cluster-wall contact time vs. solid recycle rate
4.3.2 Discussion

4.3.2.1 The effect of solid density and particle size

Figure 4.3 shows that there is a relatively direct correlation between contact time and solid density, as well as with average particle size for the same density. Although the physics governing cluster-wall contact are still uncertain, it seems intuitive that larger or heavier particles will be more difficult to move away from the wall and clusters of these particles will therefore have longer residence times. From the heat transfer model in Eq. (2.8), a longer wall residence time implies a lower heat transfer coefficient. Relating this to solid density, this would indicate that a bed running at elevated pressure (in which the particles are lighter relative to the gas) will have higher heat transfer rates as long as the gas density also plays a role. It is difficult to classify the conditions under which higher heat transfer might be observed with smaller particles, since particle size is usually unique to each CFB configuration. In general, though, higher heat transfer rates have been observed in pressurized rather than atmospheric-pressure CFBs [Basu, et al., 1996] as well as for CFBs utilizing smaller particles [see Basu & Nag, 1996, and Andersson, 1996, for example]. Before concluding that these increases in heat transfer are solely due to reduction in cluster-wall contact time, one must bear in mind that the heat transfer model requires knowledge of other hydrodynamic parameters. The fractional wall coverage by clusters and the cluster-wall contact resistance are also likely to change with operating pressure or particle size.
4.3.2.2 Probability density functions
At this time, these are the only measurements of cluster-wall contact times, so it is impossible to gather other data for comparison. With additional data, one might be able to generalize the results and postulate mechanisms that govern the cluster-wall interaction. For now, though, the nature of the observed probability distributions (Figs. 3.6 and 3.7) can be considered. A Poisson distribution is used to describe events occurring randomly in time or space, and a gamma distribution with the parameter $n$ describes the amount of time or space between every $n$ Poisson events [Lapin, 1983]. Fig. 3.7 shows that $n = 2$ fits the cluster-wall contact interaction, so that the contact time observed may be occurring between two random events. Considering the turbulence in the bulk flow to be a source of random events, and thinking of the contact times as occurring between a deposition and shedding event, then the probabilistic results might imply that the deposition and shedding of clusters at the wall occur because of turbulent motion. Again, though, the amount of data available and the measurement error does not allow for firm conclusions. Other concepts that might eliminate the “turbulent motion” theory as a possibility are discussed in the next chapter.

4.3.2.3 The effect of wall geometry
The walls in many CFB boilers are not flat; in fact, the tubes through which steam flows are often partially embedded in the walls of the riser. This type of construction is referred to as a "membrane wall", where the surface of the CFB wall has a cross-section of alternating half-tubes and fins, as in Figure 4.7.

![Diagram of CFB water wall construction](image)

**FIGURE 4.7:** Typical CFB water wall construction
Some researchers have found that the local solid-gas hydrodynamics differ near a membrane wall compared to a flat wall. Specifically, clusters appear to move more slowly and are concentrated more densely [see Zhang, et al., 1995, or Zhou, et al., 1996, for example], giving the appearance that clusters moving along the finned part of the surface are somewhat protected from the core flow by the protruding tubes. One might expect cluster-wall contact times to be longer in duration as a result, and one would also expect a corresponding drop in bed-to-wall heat transfer. Lower heat transfer rates have been observed in the fin region compared to the crest region [Andersson & Leckner, 1993], but a back-to-back study of heat transfer from flat walls and membrane walls has yet to be performed.

In order to investigate the hydrodynamic phenomena that might contribute to differences in heat transfer between flat and membrane walls, the TIV test section was modified to represent the geometry depicted in Fig. 4.7. Metal half-cylinders were attached to the heater in the TIV test section, and the LDPE wall was thermally formed to the desired shape: the 0.16 m wide wall has 3 half-tubes, the tubes having radii of 1.27 cm with centers separated by 5.08 cm. The simulated membrane wall configuration was run under conditions similar to Case 15 in Table 4.1, using the 182 μm sand particles, with \( u_0 = 3.0 \) m/s and \( G_s = 15 \) kg/m\(^2\)-s (and consequently the same average local solid concentration). The measured results show that the average contact time is about 0.20 s, representing a reduction of about 30-50% compared to the flat wall case. This is not the result that was initially expected, since a lower contact time results in a higher heat transfer coefficient. However, it was also observed that about 20% fewer clusters flowed along the wall, with virtually all of them in the finned section and very little material on the crest of the tubes. Given the geometry of the finned-tube section, this indicates that the overall fraction of the wall covered by clusters is actually about 70% lower than for the flat wall. This reduction in \( f \) would roughly predict a 50% reduction in total convective heat transfer, since the gas-convective contribution is typically much lower than the particle-convective contribution. However, there was some difficulty ensuring a smooth transition from the heater surface to the LDPE wall with the membrane-wall geometry in these experiments; it is distinctly possible that the bumpy surface contributed to the reduction in the average cluster-wall contact time. A better test-section design would be required in order to more accurately simulate the desired geometry.

The results of Lockhart, et al. [1995] are similar, showing significantly lower concentrations of particles on the crests of the tubes and therefore significantly lower heat transfer coefficients on the tubes as well. They found local heat transfer coefficients in the fin region to be 50-100% higher than on the crest of the tube. Since the fin region only comprises 25% of the surface area, though, the net heat transfer is much closer to the lower value measured on the crests of the tubes.
4.4 ANALYSIS OF CLUSTER MOTION AND PARTICLE CONVECTION

4.4.1 Cluster acceleration

The fact that the clusters do not travel at a constant velocity has the potential to affect conventional notions surrounding heat transfer by particle convection. When looking for a contact time to use in Eq. (2.8), it is easiest to assume that \( t \) is equal to some travel length divided by some velocity. However, the results presented in Fig. 4.1 indicate that clusters are not moving at a constant velocity during their residence near the wall. In this analyzing the effect of cluster acceleration on heat transfer, the cluster-motion model presented in Eqs. (4.1) and (4.2) will be employed since it is the one that most closely resembles the measured velocity histories in Fig. 4.1.

4.4.1.1 Cluster-averaged local particle-convective heat transfer

In theory, it is nice to deal with the concept of \( t \), the contact time of a cluster with the wall of a CFB, as in Eq. (2.8). In practice, however, it is inconvenient to deal with temporal coordinates, since it is impractical and perhaps impossible to re-scale the physical dimensions of a CFB relative to each cluster. Fang, et al. [1995] derived the following formula for calculating a cluster-averaged local particle-convective heat transfer coefficient:

\[
h(x) = \frac{\int_{-\infty}^{\infty} p(s) \, ds \, dx' + \int_{0}^{\infty} h_t \left( \frac{x-x'}{u_{cl}} \right) \int_{x-x'}^{\infty} p(s) \, ds \, dx'}{\int_{-\infty}^{\infty} \int_{x-x'}^{\infty} p(s) \, ds \, dx'}
\]

where \( x_0 \) is the adiabatic starting length, and the function \( h_t(t) \) is given by:

\[
h_t(t) = \left( \frac{R_{contact} \pi t}{(k \rho c_{cluster})} \right)^{-1}
\]

similar to \( h_{pc} \) in Eq. (2.7). Fang non-dimensionalized the results by defining \( Nu(x) = h(x) \cdot R_{contact} \) and a parameter \( K \) which is a function of cluster properties, and they were able to match measured heat-transfer data from Wu, et al. [1989] reasonably well. This method relied on two assumptions. One was that there was a gamma probability distribution for cluster contact times, which has been verified in the current experiments. The other was that the contact time can be replaced by a characteristic length and velocity, and the acceleration of clusters revealed in Fig. 4.1 shows that the use of a single velocity may be incorrect. Their analysis has proven to be quite
useful, however, as it is the first accurate transformation of the time-dependent and cluster-oriented particle convection model to real spatial coordinates.

4.4.1.2 Contact time as a function of distance for accelerating clusters

Fang's analysis can be modified to account for cluster acceleration by transforming the amount of time traveled into a function of the distance traveled; this relationship will not be linear because of the acceleration of the clusters. By assuming that Eq. (4.2) accurately represents the velocity history of the clusters, the function \( x(t) \) can be solved for since \( u(t) = dx/dt \). The function \( t(x) \) can also be found, although this is more difficult. Before continuing, the problem can be non-dimensionalized by defining a dimensionless distance, \( \zeta \), and a dimensionless time, \( \eta \):

\[
\zeta = \frac{x}{u_{cl}^2}; \quad \eta = \frac{g}{u_{cl}} t
\]

(4.6)

where \( u_{cl} \) represents the terminal velocity of the clusters. With these definitions, the solution for \( \zeta(\eta) \), or the dimensionless result of integrating Eq. (4.2), is:

\[
\zeta = \eta - [1 - \exp(-\eta)]
\]

(4.7)

and this function is plotted in Figure 4.8.

---

![Graph](image)

**FIGURE 4.8:** Dimensionless trajectory for an accelerating cluster
By inspection of Eq. (4.7), a direct solution for the function $\eta(\xi)$ is not possible; however, using expansion approximations and curve-fitting, the following piece-wise function $\eta(\xi)$ can be defined:

$$
\eta = \begin{cases} 
  \sqrt{2} \xi, & \xi \leq 0.05 \\
  1.82 \xi^{0.58}, & 0.05 < \xi < 1.25 \\
  \xi + 1, & \xi \geq 1.25 
\end{cases}
$$  

(4.8)

This function is superimposed on Fig. 4.8 and there are slight discontinuities at the transition points, but the maximum deviation of Eq. (4.8) from Eq. (4.7) is 5%.

4.4.1.3 Cluster-averaged particle convection, accounting for acceleration

With Eq. (4.8), the dimensionless heat-transfer coefficient as a function of the dimensionless distance along the wall can be found by reformulating Eq. (4.4) as follows:

$$
Nu(\xi) = \frac{\int_{-\zeta_o}^{0} Nu_t(\eta(\xi - \zeta')) \int_{\eta(\xi - \zeta)}^{\infty} p(\eta) \, d\eta \, d\zeta'}{\int_{-\zeta_o}^{\zeta} \int_{\eta(\xi - \zeta)}^{\infty} p(\eta) \, d\eta \, d\zeta'} + \frac{\int_{0}^{\zeta} Nu_t(\eta(\xi - \zeta')) \int_{\eta(\xi - \zeta)}^{\infty} p(\eta) \, d\eta \, d\zeta'}{\int_{-\zeta_o}^{\zeta} \int_{\eta(\xi - \zeta)}^{\infty} p(\eta) \, d\eta \, d\zeta'}$

(4.9)

where $Nu_t(\eta(\xi))$ represents the particle-convective Nusselt number as a function of time, which is a function of distance. Also, the following definitions hold:

$$
Nu_t(\eta) = \left(1 + K \cdot \sqrt{\frac{\eta}{\eta_{avg}}} \right)^{-1}, \quad K = \frac{1}{R_{contact}} \cdot \sqrt{\frac{\pi u c \eta_{avg}}{g (k p c)_{cluster}}}
$$  

(4.10)
and \( p(\eta) \) is the gamma probability distribution described in Eq. (3.10). The only substantial modification to Fang's analysis is that now the particle-convective heat transfer is calculated based on time as a non-linear function of distance, the non-linearity having been introduced as a result of cluster acceleration.

With Eq. (4.9), there are now three parameters that can be varied: \( \eta_{\text{avg}} \), the dimensionless average contact time; \( K \), the cluster-property parameter; and \( \zeta_0 \), the adiabatic starting length. The solutions for \( Nu(\zeta) \) for two values of \( \eta_{\text{avg}} \) are shown in Figure 4.9 along with the solutions using Fang's analysis; the modifications to Fang's analysis have little to no effect on \( Nu(\zeta) \) for variations in \( K \) and \( \zeta_0 \) for a given value of \( \eta_{\text{avg}} \) so only single values for \( K \) and \( \zeta_0 \) are shown.

![Figure 4.9: Comparison of Nu predictions with and without cluster acceleration](image)

As can be seen in Fig. 4.9, there can be significant error in assuming that the clusters travel at a constant velocity when the average contact time is relatively low. Fang's analysis will theoretically over-predict the cluster contribution to local heat-transfer rates by about 50% if the average cluster-wall contact times are short enough (i.e. when \( \eta_{\text{avg}} = 1 \)); this result is supported by the fact that the non-linearity in the distance-time relationship is greatest at small times, as seen in Fig. 4.8. For the
results presented for steel particles in Figs. 4.3-4.6, \( \eta_{av} = 3 \). Fang's analysis would be off by 30% in this case, so that the effect of cluster acceleration on heat transfer cannot be neglected.

4.4.2 The distribution of contact times and the development region
Prior to the analysis developed by Fang, it was usually assumed that the particle-convective heat transfer rate would decrease monotonically as the length of the heat-transfer surface increased. At some point on the surface when the length from the top exceeds that average contact distance, however, the probabilistic contact model predicts a departure from the simplistic model as the shedding and deposition of clusters becomes "balanced" or "developed". Burki, et al. [1993] were one of the first groups to measure this developing region for particle convection in detail, and Fang, et al. showed that the probabilistic contact model matched actual measurements.

When the shedding and deposition processes become developed, the heat-transfer coefficient approaches an asymptotic or "fully-developed" rate of particle-convective heat transfer, rather than continuing to decrease. That is, the heat transfer rate approaches a constant value independent of the distance along the wall. This behavior can be predicted using the probabilistic contact models of Eqs. (4.4) or (4.9) assuming that the other hydrodynamic parameters remain constant along the wall, and the developing behavior is depicted in Figure 4.10.

![Figure 4.10: Thermal development of particle-convective heat transfer, as predicted by probabilistic contact model](image)

---

FIGURE 4.10: Thermal development of particle-convective heat transfer, as predicted by probabilistic contact model
The "fully-developed" particle-convective heat-transfer coefficient can be obtained by modifying the simplistic particle-convection model. For a given \((j)\)th cluster, the time-averaged heat-transfer coefficient is:

\[
h_j = \sqrt{\frac{4 (k \rho c)_{\text{cluster}}}{\pi t_j}}
\]  

\((4.11)\)

For a population of clusters that have reached the development point, the time- and cluster-averaged heat-transfer coefficient is given by:

\[
h_{\text{av}} = \sqrt{\frac{4 (k \rho c)_{\text{cluster}}}{\pi}} \left[ \frac{\int_0^\infty t^{1/2} p(t) \, dt}{\int_0^\infty t p(t) \, dt} \right]
\]  

\((4.12)\)

where it is assumed that all clusters have the same properties and \(h_{\text{av}}\) is found by time-averaging the total energy transferred by the population of clusters. Since \(h_{\text{av}}\) is the "fully-developed" cluster heat-transfer coefficient, \(t_{\text{av}}\) can be defined as the time value appropriate for calculating \(h_{\text{av}}\):

\[
h_{\text{av}} = \sqrt{\frac{4 (k \rho c)_{\text{cluster}}}{\pi t_{\text{av}}}}
\]  

\((4.13)\)

such that:

\[
t_{\text{av}} = \left[ \frac{\int_0^\infty t p(t) \, dt}{\int_0^\infty t^{1/2} p(t) \, dt} \right]^2
\]  

\((4.14)\)

To obtain the "fully-developed" particle-convective heat-transfer coefficient, \(1/h_{\text{av}}\) is combined in series with \(R_{\text{contact}}\), as in Eq. (2.7). Furthermore, for a gamma probability distribution with \(n = 2\), it can be shown that \(t_{\text{av}}\) is related to the average contact time by:

\[
t_{\text{av}} = \frac{64}{18 \pi} t_{\text{avg}} = 1.13 \, t_{\text{avg}}
\]  

\((4.15)\)
where the correction to $t_{avg}$ is required to account for the effect of the distribution of contact times on the functional relation $h = 1/\sqrt{t}$. Therefore, the simplifications in Eqs. (4.13) and (4.15) allow one to bypass the complex probabilistic model of Eqs. (4.4) or (4.9) in order to calculate the local heat-transfer coefficient relatively far along a given surface. Wu, et al. [1989] presents one of the few measurements of local heat transfer coefficients on a relatively long surface. Using the relations presented here for contact time and other hydrodynamic parameters, Eqs. (4.13) and (4.15) can be used to predict $h_{\infty} = 60-70$ W/m²-K as measured by Wu.
4.5 SUMMARY OF EXPERIMENTAL RESULTS AND OBSERVATIONS

Through experiments and subsequent analysis, a number of characteristics have been revealed that relate to cluster motion and particle-convective heat transfer at the wall of a circulating fluidized bed. Among them are the following:

- The clusters near the wall accelerate as if in gravitational free-fall, verifying a previously postulated model, and the clusters appear to approach a terminal velocity.
- The terminal velocities of clusters at the wall of a CFB, for the given bed geometry and bed materials, are about 1 m/s and are independent of operating conditions, similar to other observations.
- The acceleration of clusters cannot be ignored when calculating heat transfer based on the particle-convective model, especially if the average contact time is short; ignoring this effect can result in the over-prediction of the cluster contribution to heat-transfer rates.
- The contact times of clusters with the smooth, plane wall of a CFB, for a given bed geometry, are between 0.20 and 0.50 s, with some variation observed with particle size and material density. These contact times can be used to predict convective heat transfer rates that are consistent with those typical of CFBs.
- The measurements of contact times follow a probability density distribution known as the gamma distribution, where the defining parameter is \( n = 2 \).
- The probabilistic contact behavior gives rise to a region in which the particle-convective heat transfer is developing, such that beyond this region the dependence on \( t \) is eliminated, resulting in a parameter \( t_{\infty} \) that can be defined to calculate the cluster contribution to heat transfer in the "fully-developed" region.
- The average cluster-wall contact time appears to be lower on membrane walls compared to flat walls; in addition, the fractional wall coverage also appears to be significantly lower. These results predict a net reduction in convective heat transfer and are consistent with other observations. However, there may be some disturbances in the experimental apparatus that lower the confidence in these results.

Given these results, the next step is to try to model some of the observed phenomena in order to better understand the results and be able to generalize them for a wider variety of circumstances. In particular, it seems logical to focus on the forces acting on a cluster near the wall. The results relating to cluster velocities indicate that there may be substantial drag forces due to the cluster-wall interaction, and if this is true, there may be lifting forces present which govern the movement of clusters away from the wall and therefore govern the cluster-wall contact time. Modeling such behavior should yield insight into the small-scale hydrodynamics at the wall of a CFB and provide a stepping stone to making Eq. (2.8) a predictive model for heat transfer in CFBs.
4.6 REFERENCES


5. MODELING: RESULTS AND DISCUSSION

5.1 OVERVIEW
Given the experimental results obtained, one basic question lingers - what is really happening near the wall? More specifically, what forces or mechanisms act on the descending clusters that cause them to move downward at certain speeds and move away from the wall at certain times? Postulating models for the physical behavior and then comparing the results of these models to the experimental results will show whether or not the models are feasible; if the results from the model match those of the experiments, then many experiments' worth of physical insight is gained.

5.1.1 The phenomena to be modeled
The observations that have commanded attention for modeling are these: that cluster velocities appear to be independent of the superficial gas velocity; that the cluster velocities are significantly slower than predicted terminal velocities, even though clusters appear to have reached a terminal velocity; and that the cluster-wall contact time appears to depend on particle density and size.

5.1.1.1 Cluster velocity and gas superficial velocity
The independence of cluster velocity from gas superficial velocity can be explained fairly simply. The clusters travel very close to the wall, perhaps within 100 μm, as shown by Lints and Glicksman [1993]. This proximity can be shown to be well within the hydrodynamic boundary layer at the wall based on single-phase fluid flow correlations [see White, 1991, e.g.]. For the gas-flow conditions in this CFB, for example, the local velocity 100 μm from the wall is 0.2 m/s when the gas superficial velocity is 3 m/s. Furthermore, it has been shown that the presence of a significant concentration of particles, as in a CFB, retards the development of the boundary layer, creating an even larger zone of low gas momentum [Rashidi, et al., 1990]. With little gas momentum available to oppose the motion of the clusters, changes in gas velocity also have relatively little effect on the local free-stream velocity. Therefore, the lack of dependence of cluster velocity on superficial gas velocity can be explained by the fact that the clusters are traveling in a region of very low gas flow.
5.1.1.2 Cluster terminal velocities relative to aerodynamic drag

As discussed earlier, a number of researchers have measured cluster descent velocities and found them to be between 0.5 and 2.5 m/s; the results of 1-1.5 m/s observed here are no different. A quick analysis of the terminal velocity using aerodynamic drag calculations will predict a significantly higher terminal velocity for a solid body far from a wall with the equivalent density as the cluster. A force balance yields the following equation for the terminal velocity of a spherical cluster:

\[
 u_{cl} = \sqrt{\frac{4}{3} \frac{\rho_s (1 - \varepsilon_{cl}) D_{cl} g}{c_d \rho_f}}.
\]  

(5.1)

where \( D_{cl} \) is the approximate diameter of the cluster and \( c_d \) is the drag coefficient for flow over a sphere [see White, 1991, e.g.]. Assuming a cluster of steel particles to be a sphere of 1 cm diameter with a solid concentration of 10%, and using \( c_d = 0.5 \) for higher Reynolds numbers, Eq. (5.1) yields a cluster velocity of about 12 m/s, which is significantly higher than that observed in any CFB. Even for a cluster of sand particles, with about one-third the density of steel, Eq. (5.1) yields a terminal velocity of about 7 m/s, which is also significantly higher than the observations. These results indicate that there must be additional forces acting on a cluster to slow its descent, and a well-postulated model might be able to shed light on the nature of these forces. It could be that the added force is a result of a direct frictional interaction between the particles in the cluster and the wall. However, it has been shown experimentally that the clusters are not in direct contact with the wall [Lints & Glicksman, 1993] and also that the roughness of the wall has no effect on the cluster descent velocities [Zhou, et al., 1996]. As a result, the additional drag forces are expected to be caused by the permeability of the cluster and the distortion of the aerodynamic flow field caused by the proximity of the cluster to the wall.

5.1.1.3 Cluster-wall contact times

There is no prior work available for understanding the mechanisms governing cluster-wall contact times. Based on the results obtained, it appears that the inertia of the bed material – particle size and material density – affect the contact time, but that no other parameters play a significant role. The fact that contact times follow a gamma probability distribution seems to indicate that random events, such as gas-phase turbulent motion, are responsible for cluster motion to and from the wall. However, the fact that particle inertia is important seems to indicate that the observed behavior may simply be a result of lateral or “lifting” forces acting to move the cluster away from the wall. Both of these scenarios will be explored.
5.1.2 **Modeling methodology**

The modeling portion of the thesis is divided into two main sections: one in which analytical modeling techniques are used, providing some fairly simple results; and one in which computational modeling techniques are used, providing results that are perhaps more accurate but more complex. The models will be presented and evaluated with respect to the data obtained, and the more feasible models will be used to explain the experimental observations.
5.2 ANALYTICAL MODELING

5.2.1 Cluster-wall contact times – relation to turbulent motion

The fact that the cluster-wall contact times follow a gamma probability distribution for a given operating condition seems to indicate that some random event or events governs the contact phenomenon, since gamma distributions are indicative of behavior between random events [Lapin, 1983]. The most significant "random event" that comes to mind in relatively high-speed channel flow (as in a CFB) is the motion of turbulent eddies. The Reynolds number in this experimental system, based on gas superficial velocity and bed diameter, is about 30,000 – clearly in the turbulent regime. As a result, it is possible that the time scales on which turbulent motion occurs are related to the time scales of cluster motion to and from the wall. By analyzing these time scales, the physical basis for the experimental observations might become apparent.

5.2.1.1 Turbulent bursts

In the concept of "turbulent bursts", it is said that turbulent eddies originate at the boundary of a flow field and build to some critical size before "bursting" away from the wall [Hinze, 1959]. It is quite possible that this burst is the disruption that causes clusters to vanish from the wall of a CFB. Previously, Hewitt [1996] discussed the correlation of local heat transfer with turbulent motion at the wall of a two-phase flow system, and Rashidi, et al. [1990] observed the disruption of solid particles at the wall in a flow field resulting from such turbulent activity. In Rashidi's experiments, however, the relative loading of solid particles was about one-tenth that found in this or typical CFB boiler systems.

5.2.1.2 The Kolmogorov time scale in turbulent flow

The time scale for eddy motion is a function of the viscosity of the fluid and the specific power required for the fluid flow [Hinze, 1959]:

$$t_k = \left( \frac{\nu}{\psi} \right)^{1/2}$$

(5.2)

where $\nu$ is the kinematic viscosity of the fluid and $\psi$ is the specific power input. To a first approximation, the viscosity of the mixture can be assumed to be that of air. The total power required for the system is equal to the pressure drop in the system multiplied by the volume flow rate of air. The total mass in the system – mostly solid particles – is equal to the weight of the particles, which is simply the total pressure drop multiplied by the riser cross-sectional area and divided by the gravitational constant. This results in the following expression for specific power input:
\[ \psi = \frac{\text{power}}{\text{mass}} = u_0 g \]  
(5.3)

which when substituted into Eq. (5.2) yields the following expression for the turbulent time scale:

\[ t_k = \left( \frac{\psi}{u_0 g} \right)^{1/2} \]  
(5.4)

The minimum fluidization velocity can be substituted in for the viscosity and other parameters through a correlation given by Grace [1982]:

\[ u_{mf} = \frac{\rho_s g d_p^2}{\mu} \]  
(5.5)

so that Eq. (5.4) becomes:

\[ t_k = \left( \frac{\rho_s g d_p^2}{u_0 u_{mf}} \right)^{1/2} \]  
(5.6)

Rearranging terms to put this in dimensionless form:

\[ t^* = t_k \sqrt{\frac{g}{D}} = \left( \frac{\rho_s u^* d_p^2}{Fr_D D^2} \right)^{1/2} \]  
(5.7)

From Eqs. (5.6) and (5.7), it becomes apparent that if the cluster-wall contact was governed by turbulent motion, then the measured contact times would depend on the solid density, particle size and gas superficial velocity. Dependence on the first two parameters has been observed, but there is no apparent dependence on the gas superficial velocity.

5.2.1.3 Comparison of the turbulent-motion model to experimental results

The fact that the model in Eq. (5.7) predicts a correlation between cluster-wall contact time and gas superficial velocity while none has been observed has one of two implications. One is that the experiments have somehow obscured the inter-relation; given the relatively large experimental error (≈ 35%), this may be the case. The other is that the turbulence model proposed is invalid; this may very well be true, but the observed distribution of contact times suggests that it is worth considering. However, other researchers have shown that the presence of relatively small particles in a turbulent flow tends to damp out the intensity of the turbulence [see Gore & Crowe, 1989, or Hetsroni, 1989, e.g.]. As a result, the presence of turbulent eddies may be significantly lower than calculated for an equivalent single-phase flow, indicating that the "turbulent burst" model for cluster-wall contact may be physically incorrect.
5.2.2 Cluster descent velocities

The analytical model proposed for describing the observed cluster velocities extends the aerodynamic analysis of Eq. (5.1) by considering additional forces due to the flow of air through the cluster. The effect of the wall is ignored for now, since that adds a level of complexity that drives the problem beyond simple analytical methods.

5.2.2.1 Modeling assumptions

This model considers the opposing gas flow to be negligible, so that gas velocity considered is only due to the relative motion of the cluster. Furthermore, only terminal velocities are considered, where the opposing forces can be divided into two categories: aerodynamic drag, due to the flow around the cluster, which is usually divided further into form and skin-friction drag; and permeable drag, or drag due to flow through the cluster. Furthermore, it is assumed that a single length-scale can be used to describe the cluster. The variable $D_{cl}$ will be used to represent this length scale, where $D_{cl}^2$ represents the cross-sectional area and $D_{cl}^3$ represents the volume of the cluster. The variable $u_{cl}$ is used to represent the terminal velocity of the cluster, as shown in Figure 5.1.

\[
\begin{align*}
\text{cluster} & \quad \text{permeable drag force} \\
\text{aerodynamic drag force} & \\
\text{\[u_{gas} = 0\]} & \\
\end{align*}
\]

FIGURE 5.1: Schematic of simple cluster-descent model
Any full-blown analysis of this system is very complex – either to model, since the flows through and around the clusters are coupled, or to arrive at a solution, since the equations involved are nonlinear. As a result, it is more illustrative to look at the extremes to try to predict cluster velocities. These extremes are: all of the gas flows through the cluster or all of the gas flows around the cluster. In either extreme, the analysis simplifies because only one equation needs to be solved.

5.2.2.2 Drag resulting from permeability only
The Ergun equation [Ergun, 1952] can be used to predict the forces on a permeable cluster (although the Ergun equation may not be valid for solid fractions as low as those expected):

\[
\frac{\Delta P}{L} = 150 \frac{(1 - \varepsilon_{cl})^2}{\varepsilon_{cl}^3} \frac{\mu}{d_p^2} u_1 + 1.75 \frac{(1 - \varepsilon_{cl})}{\varepsilon_{cl}^3} \frac{\rho_f}{d_p} u_1^2 \tag{5.8}
\]

where \( u_1 \) denotes the cluster velocity predicted when all of the flow passes through the cluster and none of the flow diverts around it. The Ergun equation normally depends on the sphericity of the particles as well, but this parameter can be assumed to be unity for a first approximation. Given that \( L \) in Eq. (5.8) is the same as \( D_{cl} \), the length scale of the cluster, the drag force on the cluster is given by the pressure drop acting on the projected area of the cluster:

\[
F_d = 150 (1 - \varepsilon_{cl}) \frac{\mu D_{cl}^3}{\varepsilon_{cl}^3} u_1 + 1.75 \frac{(1 - \varepsilon_{cl})}{\varepsilon_{cl}^3} \frac{\rho_f D_{cl}^3}{d_p} u_1^2 \tag{5.9}
\]

The weight of the cluster is given by:

\[
W_{cl} = m_{cl} g = \rho_s (1 - \varepsilon_{cl}) D_{cl}^3 g \tag{5.10}
\]

so that equating the weight of the cluster and the drag on the cluster yields the following expression for the terminal velocity:

\[
150 \frac{(1 - \varepsilon_{cl})^2}{\varepsilon_{cl}^3} \frac{\mu D_{cl}^3}{d_p^2} u_1 + 1.75 \frac{(1 - \varepsilon_{cl})}{\varepsilon_{cl}^3} \frac{\rho_f D_{cl}^3}{d_p} u_1^2 = \rho_s (1 - \varepsilon_{cl}) D_{cl}^3 g \tag{5.11}
\]

Canceling terms in Eq. (5.11) results in the following expression:

\[
150 \frac{(1 - \varepsilon_{cl})}{\varepsilon_{cl}^3} \frac{\mu}{d_p^2} u_1 + 1.75 \frac{\rho_f}{\varepsilon_{cl}^3} u_1^2 = \rho_s g \tag{5.12}
\]
Simplifying even further, let the coefficients on the left-hand side of Eq. (5.12) be replaced by \( C_v \) and \( C_i \), respectively (for viscous and inertial coefficients), such that they are functions only of the void fraction of the cluster:

\[
C_v = 150 \left( \frac{1 - \varepsilon_{cl}}{\varepsilon_{cl}^3} \right) \quad \text{and} \quad C_i = 1.75 \frac{1}{\varepsilon_{cl}^3} \tag{5.13}
\]

Using these, Eq. (5.12) can be re-written as:

\[
C_v \frac{\mu}{d_p^2} u_1 + C_i \frac{\rho_f}{d_p} u_1^2 = \rho_s g
\]

or rearranging terms:

\[
u_1^2 + \frac{C_v \mu}{C_i \rho_f d_p} u_1 - \frac{1}{C_i} \frac{\rho_s g d_p}{\rho_f} = 0 \tag{5.15}
\]

and a solution for \( u_1 \) can be obtained by applying the quadratic formula to Eq. (5.15):

\[
u_1 = \frac{1}{2} \left[ -\left( \frac{C_v \mu}{C_i \rho_f d_p} \right) + \sqrt{\left( \frac{C_v \mu}{C_i \rho_f d_p} \right)^2 + \frac{4 \rho_s g d_p}{\rho_f}} \right] \tag{5.16}
\]

Non-dimensionalizing this result by \( u_{mf} \) yields:

\[
u_1* \equiv \frac{u_1}{u_{mf}} = \frac{1}{2} \left[ -\left( \frac{C_v \mu}{C_i \rho_f d_p u_{mf}} \right) + \sqrt{\left( \frac{C_v \mu}{C_i \rho_f d_p u_{mf}} \right)^2 + \frac{4 \rho_s g d_p}{\rho_f u_{mf}^2}} \right] \tag{5.17}
\]

\( u_{mf} \) can be replaced by the following approximation from Grace [1982]:

\[
u_{mf} = C_{mf} \frac{\rho_s g d_p^2}{\mu} \tag{5.18}
\]

where \( C_{mf} = 0.00075 \). Substituting Eq. (5.18) into Eq. (5.17) and rearranging terms yields:

\[
u_1* = \frac{1}{2} \left[ -\left( \frac{C_v \mu}{C_i C_{mf} A_r} \right) + \sqrt{\left( \frac{C_v \mu}{C_i C_{mf} A_r} \right)^2 + \frac{4 \rho_s g d_p}{\rho_f C_{mf}^2 A_r}} \right] \tag{5.19}
\]

where the \( A_r \) is the Archimedes number, defined by:

\[
A_r \equiv \frac{(\rho_s - \rho_f) \rho_s g d_p^3}{\mu^2} = \frac{\rho_s \rho_f g d_p^3}{\mu^2} \tag{5.20}
\]
Except for two constants \((C_v, C_l)\) that depend on the solid concentration in the cluster, the only parameter governing the cluster velocity in this analysis is the Archimedes number. Given that a typical solid fraction in a cluster is about 10\% [Lints & Glicksman, 1993], an "order-of-magnitude" expression for the dimensionless cluster velocity can be obtained:

\[
    u_{1*} = \frac{1}{2} \left[ \frac{10^4}{A_r} + \sqrt{\frac{10^8}{A_r^2} + \frac{3 \times 10^6}{A_r}} \right] \tag{5.21}
\]

In the limit when \(A_r \gg 100\), the right-hand term under the square-root dominates; in a CFB, a typical value for Archimedes number might be \(A_r \geq 100\). As a result, Eq. (5.21) simplifies to:

\[
    u_{1*} = \frac{900}{\sqrt{A_r}} \tag{5.22}
\]

This is an expression for the terminal velocity of a cluster that depends on the Archimedes number, assuming that the only drag force acting on the cluster is that due to the air flowing through the cluster.

### 5.2.2.3 Drag resulting from aerodynamics only

Assuming that all of the flow goes around the cluster, the drag force is given by:

\[
    F_d = \frac{1}{2} c_d \rho_f u_2^2 D_{cl}^2 \tag{5.23}
\]

where \(c_d\) is the drag coefficient, for flow around a sphere perhaps, and \(u_2\) represents the velocity of the cluster when analyzed for aerodynamic drag only. Equating this to the weight of the cluster as given in Eq. (5.10) results in the following force balance:

\[
    \frac{1}{2} c_d \rho_f u_2^2 D_{cl}^2 = \rho_s (1 - \epsilon_{cl}) D_{cl}^3 g \tag{5.24}
\]

which can be rearranged to provide a solution for \(u_2\):

\[
    u_2 = \sqrt{\frac{2 \rho_s}{c_d \rho_f} (1 - \epsilon_{cl}) D_{cl} g} \tag{5.25}
\]

Non-dimensionalizing this result by \(u_{mf}\) as in the earlier analysis yields:

\[
    u_{2*} = \frac{u_2}{u_{mf}} = \sqrt{\frac{2 \rho_s (1 - \epsilon_{cl}) D_{cl} g}{c_d \rho_f u_{mf}^2}} \tag{5.26}
\]

Substituting Eq. (5.18) for \(u_{mf}\) and regrouping variables to form an Archimedes number results in the following expression for the dimensionless cluster velocity:
\[ u_{2*} = \frac{1}{C_{mf}} \frac{1}{\sqrt{Ar}} \sqrt{\frac{2}{\varepsilon_d} \frac{D_{cl}}{d_p} (1 - \varepsilon_{cl})} \] (5.27)

Performing an "order-of-magnitude" analysis similar to that used to derive Eq. (5.22), several assumptions can be made. For example: the cluster solid fraction is typically about 10\%, the drag coefficient for a sphere is on the order of unity, a typical cluster length scale is about 1 cm, and a typical particle size in a CFB is about 100 \( \mu m \). Again, each of these numbers are only good within an order of magnitude, but the assumptions result in the following simplification to Eq. (5.27):

\[ u_{2*} = \frac{6 \times 10^3}{\sqrt{Ar}} \] (5.28)

The functional dependence of cluster velocity on Archimedes number is exactly the same as that given in Eq. (5.22), and the constants of proportionality are of similar magnitude. In other words, whether considering drag forces caused by flow through the cluster or flow around the cluster, the result differs only by a constant of proportionality. From this, one could conclude that the functional dependence will be the same if both drag forces act on the cluster.

5.2.2.4 Comparison of model to experimental results

The preceding analysis suggests that the terminal velocity of a cluster, when non-dimensionalized by the minimum fluidization velocity of the particles, can be correlated to the square-root of the Archimedes number of the gas-solid system. In order to test this analysis, a large number of cluster-velocity measurements have been collected from the available literature on CFBs. These measurements and the conditions under which they were made are summarized in Table 5.1; all of the data were obtained under atmospheric-pressure conditions. The data presented in Table 5.1 can be plotted in the form of \( u_{cl}/u_{mf} \) vs. \( 1/\sqrt{Ar} \) – the same form derived in Eqs. (5.22) and (5.28).

Figure 5.2 presents the data in this form, along with a line representing the function \( u_* = 1000/\sqrt{Ar} \) which, by inspection, provides a reasonably good curve fit to most of the data. The fact that the constant of proportionality is 1000 bodes well for the preceding analysis, since the constants found in the two extremes were of that order of magnitude. All of the data appear to follow the trend line fairly well; only the data of Rhodes, Zhang and Yang deviate appreciably. It seems that for the relatively small amount of effort expended, a fairly decent correlation for cluster velocity has been determined.
<table>
<thead>
<tr>
<th>Authors</th>
<th>bed temp. (K)</th>
<th>bed mat'l.</th>
<th>$d_p$ (μm)</th>
<th>$\rho_r$ (kg/m$^3$)</th>
<th>Ar</th>
<th>calc. $u_{mf}$ (cm/s)</th>
<th>meas. $u_{cl}$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rhodes, et al. [1992]</td>
<td>300</td>
<td>alum.</td>
<td>75</td>
<td>2460</td>
<td>33</td>
<td>0.54</td>
<td>0.3</td>
</tr>
<tr>
<td>Zhou, et al. [1995]</td>
<td>300</td>
<td>sand</td>
<td>213</td>
<td>2640</td>
<td>816</td>
<td>4.6</td>
<td>1.5</td>
</tr>
<tr>
<td>Harris, et al. [1993]</td>
<td>300</td>
<td>FCC</td>
<td>60</td>
<td>1700</td>
<td>12</td>
<td>0.24</td>
<td>0.8</td>
</tr>
<tr>
<td>Ishii, et al. [1989]</td>
<td>300</td>
<td>FCC</td>
<td>60</td>
<td>1000</td>
<td>7</td>
<td>0.14</td>
<td>0.5</td>
</tr>
<tr>
<td>Bader, et al. [1988]</td>
<td>300</td>
<td>FCC</td>
<td>76</td>
<td>1710</td>
<td>24</td>
<td>0.38</td>
<td>0.9</td>
</tr>
<tr>
<td>this study</td>
<td>300</td>
<td>steel</td>
<td>69</td>
<td>6980</td>
<td>73</td>
<td>1.2</td>
<td>1.2</td>
</tr>
<tr>
<td>this study</td>
<td>300</td>
<td>sand</td>
<td>128</td>
<td>2660</td>
<td>178</td>
<td>1.6</td>
<td>1.2</td>
</tr>
<tr>
<td>this study</td>
<td>300</td>
<td>sand</td>
<td>182</td>
<td>2650</td>
<td>490</td>
<td>3.2</td>
<td>1.1</td>
</tr>
<tr>
<td>Hartge, et al. [1988]</td>
<td>300</td>
<td>FCC</td>
<td>85</td>
<td>1500</td>
<td>29</td>
<td>0.42</td>
<td>1.0</td>
</tr>
<tr>
<td>Hartge, et al. [1988]</td>
<td>300</td>
<td>ash</td>
<td>120</td>
<td>2600</td>
<td>144</td>
<td>1.5</td>
<td>1.0</td>
</tr>
<tr>
<td>Zhang, et al. [1995]</td>
<td>1125</td>
<td>n/a*</td>
<td>330</td>
<td>2600*</td>
<td>149</td>
<td>4.7</td>
<td>0.9</td>
</tr>
<tr>
<td>Yang &amp; Gautam [1995]</td>
<td>300</td>
<td>n/a*</td>
<td>250</td>
<td>2250</td>
<td>1080</td>
<td>5.2</td>
<td>0.4</td>
</tr>
<tr>
<td>Wang, et al. [1993]</td>
<td>300</td>
<td>sand</td>
<td>530</td>
<td>2300</td>
<td>11000</td>
<td>22.</td>
<td>2.0</td>
</tr>
<tr>
<td>Lints [1992]**</td>
<td>300</td>
<td>sand</td>
<td>80</td>
<td>2660</td>
<td>44</td>
<td>0.66</td>
<td>1.6</td>
</tr>
<tr>
<td>Lints [1992]**</td>
<td>300</td>
<td>alum.</td>
<td>600</td>
<td>2750</td>
<td>19000</td>
<td>31.5</td>
<td>1.8</td>
</tr>
<tr>
<td>Wu, et al. [1991]</td>
<td>300</td>
<td>sand</td>
<td>170</td>
<td>2650</td>
<td>425</td>
<td>3.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

* information not available, values have been estimated
** data for particles at the wall of a tube with upward gas flow, not in a real CFB

**TABLE 5.1: Measurements of cluster velocities**
FIGURE 5.2: Cluster velocities – model predictions vs. measurements
5.2.2.5 Limitations of this analysis

Even though this analysis is not exact, the data indicate that the actual behavior might very well be some combination of the behavior in the two extremes. However, there are some serious limitations that must be addressed. First, it is impossible to determine whether the flow actually goes through or around the cluster without analyzing the problem in detail. Second, in simplifying Eq. (5.21) to Eq. (5.22), it was assumed that \( Ar >> 100 \). However, the expression for \( u_{mf} \) introduced in Eq. (5.18) is best for \( Ar \leq 1000 \), so there is some inconsistency in the underlying assumptions. Third, the data points from Rhodes, Zhang and Yang deviate from the model by 100% or more (the scale in Fig. 5.2 is logarithmic, so the proximity of the points to the line can be misleading). Finally, it is known that the clusters travel very close to the wall, so that ignoring the effect of the wall on the flow field around the cluster is physically incorrect. For these reasons, more precise analysis of the flow on a cluster near a wall is required, and for this level of complexity, computational models are required.
5.3 COMPUTATIONAL MODELING
To arrive at a more complete understanding of the experimental observations, several effects need to be considered. Isolating a single cluster as the unit to be analyzed, there is flow past it, flow through it, and some effect of the wall on the flow field. The most effective way of incorporating all of these effects in the analysis is to employ a computational model. The results from this can be used to quantify the lift and drag forces acting on a cluster and determine if the mechanisms postulated actually govern cluster motion and cluster-wall contact times. This section of the thesis is structured as follows: pertinent prior research is reviewed, the computational model and methods are described, the computational model is validated for unbounded flow past a solid cylinder, the model is used to determine the effect of permeability and proximity to the wall, and finally these results are evaluated with respect to the experimental observations.

5.3.1 Review of prior research
There does not appear to be any previous work under exactly the same flow conditions to be studied here; namely, no one has ever studied the flow past a permeable body moving past a stationary wall in a quiescent fluid. There are, however, studies of flow past permeable bodies and flow past solid bodies near walls, but not both. Most of the previous studies have been conducted at relatively low Reynolds numbers, where the Reynolds number is defined based on the diameter of the body.

5.3.1.1 The effect of permeability on drag
Previous work has shown that permeability has a negligible effect on drag at "vanishingly small" Reynolds numbers in unbounded flow [see Joseph & Tao, 1964, or Neale, et al., 1973, e.g.]. Moving out of the viscous limit, Masliyah and Polikar [1980] found the permeability of a sphere to have a negligible effect on drag at Re = 10 but they found an increase in drag of about 20% at Re = 100. Since clusters in a CFB move at Reynolds numbers approaching 1000, the results of Masliyah and Polikar give some indication that permeability can play a role in reducing the terminal velocity of the clusters.

5.3.1.2 The effect of a wall on lift and drag
Goldman, et al. [1967] showed that the motion of a sphere near a wall in a quiescent fluid can increase drag by a factor of about five, with the augmentation of drag depending on distance from the wall. Saffman [1965] showed that a sphere in the boundary layer of a viscous flow near a wall will experience a force directed away from the wall, or a lifting force. Both of these results are also for "vanishingly small" Reynolds numbers, and therefore potentially lack applicability to the current configuration. At the other end of the Reynolds number spectrum, Carr [1988] tested scale
models of automobiles in a wind tunnel with $Re > 10^5$ and with a conveyor belt to simulate relative motion between the automobile and the ground. Carr's results show that the proximity of the body to the ground can either increase or decrease drag, depending on the shape of the body. Similarly inconclusive results were compiled by Hucho [1987], although it appears that drag and lift tend to increase with decreasing ground clearance for automobiles with increasing underbody convexity. These results say nothing firm about cluster motion, but they indicate that the presence of the wall should have an effect, and that lift and drag might both increase with proximity to the wall if the cross-section of the cluster is considered to be convex.

5.3.2 Description of CFD model

5.3.2.1 CFD code

A commercially-available program for computational fluid dynamics (CFD) was used in this study. The CFD program is PHOENICS, Version 2.1.1, produced by CHAM Ltd. [1991]. PHOENICS employs finite-volume methods for the solution of the governing equations for fluid dynamics, and provides as output the pressure and velocity fields on the user-specified discrete computational domain. Internal to PHOENICS are some built-in functions to assist in computations for more complex flows, accounting for the effects of turbulence or permeability, for example. The use of these is discussed later.

5.3.2.2 Computational domain and discretization

For simplicity, a cluster is modeled as a cylinder – a round, two-dimensional object. Although the results from such a model will not be exactly applicable to clusters in a CFB, the results will indicate the presence and significance of forces acting on a permeable body moving near a wall. Figure 5.3 gives a schematic of the model and the pertinent nomenclature. Two computational domains have been defined: one for the simple case of flow past a cylinder far away from any walls, the other for the study of the relative motion between the cylinder and the wall. For all cases, only steady-state conditions are considered. Appendix E contains the PHOENICS input files for the definition of both domains.
**Computational domain – unbounded flow**

For the study of unbounded flow past a cylinder, the computational domain is a polar coordinate system with the cylindrical body located at the center; an illustration of the domain is shown in Figure 5.4.
The domain was optimized by modifying the overall size of the domain and the fineness of the discretization until negligible change was seen in the results; Table 5.2 presents the results of this optimization procedure for steady-state flow past a solid cylinder.

<table>
<thead>
<tr>
<th>Domain size as a multiple of $D$</th>
<th>$c_d$ computed</th>
<th>Fineness as fraction of $D$</th>
<th>$c_d$ computed</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0.96</td>
<td>1/10</td>
<td>0.53</td>
</tr>
<tr>
<td>20</td>
<td>0.86</td>
<td>1/20</td>
<td>0.77</td>
</tr>
<tr>
<td>50</td>
<td>0.83</td>
<td>1/30</td>
<td>0.80</td>
</tr>
<tr>
<td>40</td>
<td>0.82</td>
<td>1/40</td>
<td>0.81</td>
</tr>
<tr>
<td>50</td>
<td>0.82</td>
<td>1/50</td>
<td>0.82</td>
</tr>
</tbody>
</table>

TABLE 5.2: Optimization of computational domain

The optimal domain size for this study was found to be 50 cylinder diameters. The domain itself was discretized in two distinct regions. The inner region, representing the cylindrical body, occupies one cylinder diameter, and it is divided into evenly-spaced circumferential divisions and evenly-spaced radial divisions. The outer region occupies the outer 49 cylinder diameters, and it is divided into evenly spaced circumferential divisions also, but it is divided radially in exponentially expanding elements. The radial length of the first element in the outer region is 1/50 of a cylinder diameter, and the elements double in length every 2.7183 elements. The increasing coarseness is utilized because the level of fineness required near the body is not required out in the free-stream; this discretization technique has been used previously for computational efficiency in this type of study [Nandakumar & Masliyah, 1982].

Computational domain – motion of cylinder near a wall

For the study of the motion of a cylinder relative to a wall in a quiescent fluid, the computational domain is a Cartesian coordinate system that is distorted into a round shape in the region of the cylinder. In this study, the origin of the coordinate system has been defined at the point on the cylinder that is closest to the wall. The computational domain and the discretized elements are shown in Figure 5.5.
The domain was optimized using procedures similar to the unbounded case. The domain extends three cylinder diameters upstream of the windward face of the cylinder, 10 diameters downstream of the leeward face of the cylinder, and 5 diameters above the top-most point on the cylinder. The gap between the cylinder and the wall is one of the parameters to be varied in this study. One hundred elements comprise the cylinder itself, for which the permeability can be varied.

5.3.2.3 Governing equations and boundary conditions
In the inner region, mass conservation and D'Arcy's Law for flow in a permeable body are applied in order to solve for the pressure and velocity fields. The mass conservation equation for two-dimensional, incompressible flow is given by the following expression [see White, 1992, e.g.]:

\[ \nabla \cdot \vec{v} = 0 \]  \hspace{1cm} (5.29)

and the conservation of momentum is expressed in D'Arcy's Law [see Scheidegger, 1960, for example]:

\[ \nabla P = \frac{\mu \vec{v}}{\kappa} \]  \hspace{1cm} (5.30)
In the outer region, the mass conservation expression of Eq. (5.29) is applied, while the two-dimensional, incompressible form of the Navier-Stokes equation is used to solve for the momentum of the fluid flow [see White, 1992, e.g.]:

\[ \vec{v} \cdot \nabla \vec{v} = - \nabla P + \mu \nabla^2 \vec{v} \]  

(5.31)

The solutions for the pressure and velocities are matched at the interface of the two regions, and the boundary conditions that are specified for the computation differ slightly between the unbounded and near-wall flow cases. For the unbounded case, the pressure on the exterior of the domain and the velocity on the exterior of the domain to the windward side of the cylindrical body are specified. Matching the velocity field of the outer region to that of the inner region provides the second boundary condition for velocity required by Eq. (5.31). For the near-wall case, in addition to the boundary conditions already specified, the wall is defined as a solid surface moving at the same velocity as the incoming air.

### 5.3.2.4 Turbulence modeling

The effects of turbulence were ignored for the purposes of this study. PHOENICS has a variety of self-contained turbulence models of the "k-epsilon" type. The use of these models at the Reynolds numbers in question only affected the results for drag coefficient by about 0.1%; this was not deemed sufficient to warrant the added computational time (approximately double when including turbulence) or complexity. This is likely due to the fact that turbulence does not play a significant role in the boundary layer on a cylinder for \( Re < 10,000 \); the conditions studied here represent \( 10 \leq Re \leq 1000 \). In recognition of the fact that neglecting turbulence introduces some inaccuracy, the results that are presented are based on a comparison of computed results for solid cylinders and computed results for permeable cylinders in which both results ignore the effects of turbulence. By using relative results rather than absolute results, it is expected that the inaccuracy due to this assumption will be reduced.

### 5.3.2.5 Calculation of lift and drag coefficients

As a final word on the computational methods, some discussion of the "post-processing" of the raw output is warranted. As mentioned previously, the PHOENICS output consists of the pressure and velocity fields in each discretized element. A linear momentum balance on a control volume around the cylinder yields the following expression for the net force on the body as a function of the change in momentum of the fluid passing through the control volume and the net pressure force on the control volume:
\[ F_i = \int_A P_i \, dA - \int_A \rho \, v_i \, \vec{v} \cdot d\vec{A} \]  

(5.32)

where \( i \) represents one of the directions. In the \( x \)-direction, \( F_i = F_x \) is the drag force, while in the \( y \)-direction, \( F_i = F_y \) is the lifting force. Given either component of the force on the cylinder, the drag and lift coefficients can be calculated from their definitions for a cylindrical body [see White, 1992, e.g.]:

\[ c_d = \frac{F_x}{\frac{1}{2} \rho \, u_d^2 D} \quad \text{and} \quad c_l = \frac{F_y}{\frac{1}{2} \rho \, u_d^2 D} \]  

(5.33)

where \( u_d \) is the velocity of the cluster and is the free-stream velocity in the model.

### 5.3.3 Validation of CFD model

In order to establish a benchmark for all of the computational results, the drag for a solid cylinder in unbounded flow is considered first. Figure 5.6 presents the computational results for drag coefficient vs. Reynolds number, comparing the output from this study to the wide body of empirical data for this flow configuration. The curve representing the empirical data is actually a curve fit of the body of data by Sucker and Brauer [1975]. This figure shows that the computational model is accurate within about 10% near \( Re = 1000 \), which is the range of interest; the error strays up to 30% near \( Re = 10 \). One possible source of error is the fact that the solutions obtained are for steady flow, while the wake behind a cylinder is inherently unsteady. To gauge the integrity of the computational output further, though, Figure 5.7 presents the computational results for the angle of flow separation from the cylinder vs. Reynolds number, where the separation angle is measured from the leading stagnation point. Again, the body of empirical data is represented by a curve fit determined by Sucker and Brauer [1975], and excellent agreement is seen between the computational and empirical results.

Figures 5.8, 5.9 and 5.10 show the computed streamlines for flow past a solid cylinder at \( Re = 10, 100 \) and 1000. In these figures, the qualitative trends in the expected flow patterns are seen – ranging from very little flow separation and recirculation at the lower Reynolds numbers to significant separation and recirculation at higher Reynolds numbers. The results shown in Figs. 5.6-5.10 instill a high level of confidence in the competence of the computational model for this geometry and range of Reynolds numbers.
FIGURE 5.6: Drag coefficient on a solid cylinder, computed vs. empirical

FIGURE 5.7: Separation angle for flow past a solid cylinder, computed vs. empirical
FIGURE 5.8: Computed streamlines for flow past a solid cylinder, $Re = 10$
FIGURE 5.9: Computed streamlines for flow past a solid cylinder, $Re = 100$
FIGURE 5.10: Computed streamlines for flow past a solid cylinder, $Re = 1000$
5.3.4 Drag on a porous cylinder in unbounded flow

5.3.4.1 Flow past a permeable cylinder, $Re = 10, 100, 1000$

Employing D'Arcy's Law as a permeability model for the cylinder (the inner region), the computational model can be used to determine the drag coefficient as a function of both Reynolds number and permeability. An appropriate non-dimensionalization of permeability is division by the square of the cylinder diameter; this parameter – $\kappa D^2$ – will be referred to as the "permeability ratio". In the interest of efficiency, the drag coefficient as a function of the permeability ratio was computed only for $Re = 10, 100$ and 1000. These values were chosen for their representation of distinctly different flow regimes, as seen in Figs. 5.8-5.10, as well as for the fact that these values cover two orders of magnitude in Reynolds number. $Re = 10$ represents the high end of the current state of knowledge of this type of flow. $Re = 1000$ represents a typical flow condition expected for a cluster in a CFB, given a cluster of about 1 cm in size moving at about 1.5 m/s in ambient air.

Figures 5.11, 5.12 and 5.13 present the computational results of "drag ratio" vs. permeability ratio for $Re = 10, 100$ and 1000, respectively. The results for drag are presented as a drag ratio – the ratio of drag on the porous cylinder to drag on a solid cylinder under the same conditions – in order to minimize any inaccuracies inherent in the computational model, as discussed previously.

**FIGURE 5.11:** Drag ratio vs. permeability ratio for flow past a cylinder, $Re = 10$
FIGURE 5.12: Drag ratio vs. permeability ratio for flow past a cylinder, $Re = 100$

FIGURE 5.13: Drag ratio vs. permeability ratio for flow past a cylinder, $Re = 1000$
In each of these figures, the behavior in the extremes of permeability are obvious: a drag ratio of unity for low permeability and a drag ratio of zero for high permeability. Additionally, a peak in the value of the drag ratio at intermediate values of permeability is observed with the higher Reynolds numbers, and the magnitude of this peak and the value of permeability corresponding to it appears to vary with $Re$. The behavior of drag with permeability ratio are discussed next, along with comparisons to the experimental measurements plotted in Figs. 5.11-5.13.

5.3.4.2 Relevance of drag vs. permeability results

The results for drag as a function of permeability are relevant to clusters in CFBs. Clusters in fluidized beds have been measured to have porosity levels of about 80% to 95% [Lints & Glicksm., 1993]. The porosity (or volumetric void fraction), $\varepsilon$, can be related to the permeability, $\kappa$, by the Carman-Kozeny relation [Scheidegger, 1960]:

$$\kappa = \frac{1}{180} \frac{\varepsilon^3 d_p^2}{(1 - \varepsilon)^2}$$  \hspace{1cm} (5.34)

where $d_p$ is the diameter of the particles in the fluidized bed; particles are typically about 100 $\mu$m in diameter. Therefore, applying Eq. (5.34) to a typical cluster (with $D_{cl} = 1$ cm) a permeability ratio in the range of $10^{-4}$ to $10^{-5}$ can be calculated, which is roughly where the peak increase in drag is observed. This indicates that these results are physically relevant. Approximate values for porosity, using Eq. (5.34) have been added to the scales in the figures, where it has been assumed that $d_p = 100 \mu$m and $D_{cl} = 1$ cm.

5.3.4.3 High and low permeability limits

The behavior in the extremes of high and low permeability can be explained both qualitatively and analytically. In the case of low permeability, the drag should approach that of a solid cylinder (i.e., the drag ratio should approach unity) and as can be seen in Figs. 5.11-5.13, it does. In the case of high permeability, the drag should approach zero, which is also in agreement with the computed results. It is interesting to note that the zero-drag limit is approached asymptotically, which is a result that can be predicted analytically. Figure 5.14 shows a typical set of predicted streamlines for a high permeability case; from this figure, it can be seen that the streamlines pass through the cylinder with little deviation.
FIGURE 5.14: Computed streamlines for flow past a highly permeable cylinder, $Re = 1000$, $\varepsilon > 0.99$
From this, the drag can be considered to be caused by the pressure drop across each streamline within the cylinder. Applying D'Arcy's Law for each chord of the cylinder yields the following integral:

\[
F_x = \int_{-\pi/2}^{\pi/2} \Delta P \, dA = \int_{-\pi/2}^{\pi/2} \frac{\mu \, u_{cl}}{K} \, R_c^2 \cos \theta \, d\theta
\]  

(5.35)

for which the solution for drag coefficient, applying Eq. (5.33), is:

\[
c_d = \frac{2}{Re} \left( \frac{K}{D_{cl}^2} \right)^{-1}
\]  

(5.36)

and this result shows that the drag will asymptotically approach zero as the permeability increases. Eq. (5.36) also shows a dependence on \( Re \), in that the permeability at which the drag approaches zero varies inversely with Reynolds number. This result is also seen in the computational results of Figs. 5.11-13, as the permeability for near zero-drag decreases by an order of magnitude with an order of magnitude increase in Reynolds number.

5.3.4.4 Intermediate values of permeability

For intermediate values of the permeability ratio, a peak in the drag ratio is observed at higher values of Reynolds number. For \( Re = 1000 \), this peak represents an increase in drag ratio of 46% at a permeability ratio of \( 4 \times 10^{-4} \), while for \( Re = 100 \) the magnitude of the peak is 31% at a permeability ratio of \( 3 \times 10^{-3} \). For \( Re = 10 \), no peak is observed. It is more difficult to predict this intermediate behavior than it is to predict the behavior in the extreme limits; in fact, construction of an analytical model for intermediate values of permeability has proved to be elusive. To confirm the general trends indicated by the computed results, however, there is a limited amount of experimental and computational data available in this range of Reynolds numbers.

Flow past permeable spheres, \( Re \leq 100 \)

Masliyah and Polikar [1980] conducted experiments using permeable spheres and Nandakumar and Masliyah [1982] conducted a computational study of flow past permeable spheres. To put their data in terms of a drag ratio, a curve fit for the empirical data for drag coefficients on a solid sphere developed by Turton & Levenspiel [1986] was used. Their data are superimposed in Figs. 5.11 and 5.12 (there is no data for \( Re = 1000 \)). From this, it can be seen that there is excellent agreement between their experimental and computational results and the present computational results for \( Re = 10 \). Furthermore, their data provides experimental verification that a substantial increase in drag ratio at \( Re = 100 \) does occur, although not enough data is available to reconstruct
the most interesting portions of the current computational results. These comparisons seem to
indicate that the observed peaks in drag coefficients are physically realistic.

**Flow past permeable cylinders, \( Re = 1000 \)**

Until now, there had been no data available in the literature for comparison at \( Re = 1000 \). To
remedy this, experiments were conducted on permeable cylinders in a wind tunnel in order to both
extend the range of Reynolds numbers and provide the proper geometry for validation of the
computed results. An open circuit wind tunnel (using ambient air) with a rectangular cross-section
of 91 cm by 41 cm was used for these experiments. Two cylinders were prepared, both 1.27 cm
in diameter and 30.5 cm long. One cylinder was machined from solid aluminum, while the other
was machined from Duocel® — an open-celled aluminum foam — supplied by ERG Materials and
Aerospace Corp. of Oakland, CA. According to measurements conducted here and by ERG, the
foam has a solid fraction of 7.5%, a pore density of 40 pores per inch, and a permeability of
1.07 \( \times 10^{-8} \) m². Given the diameter of the cylinder tested, a permeability ratio (\( \kappa/D^2 \)) of 6.6 \( \times 10^{-5} \)
can be calculated. To obtain a \( Re = 1000 \) in the wind tunnel, a velocity of 1.2 m/s is required.

The drag coefficient on a cylinder can be measured simply by measuring the velocity profile in the
wake of the cylinder, given a uniform velocity profile upstream of the cylinder and given that the
wake measurement is made far enough downstream to allow for pressure recovery. The following
application of the linear momentum equation can be used to find the drag coefficient from these
measurements [see Pope and Harper, 1966, e.g.]

\[
    c_d = \frac{2}{D} \int_y \left[ \frac{u(y)}{u_{\infty}} \right] \left[ \frac{u'(y)}{u_{\infty}} \right] \, dy
\]  

(5.37)

where \( u(y) \) is the velocity profile in the wake and \( u_{\infty} \) is the free-stream velocity (simulating the
velocity of the cluster). Both the free-stream velocity and the wake profile were measured using a
calibrated thermal-anemometry system, and the wake profiles were measured roughly 40 cylinder
diameters downstream of the cylindrical body.

Figure 5.15 presents the velocity profiles in the wakes of the solid cylinder and the permeable
cylinder. The free-stream velocity over the solid cylinder was 1.21 m/s with an air temperature of
19°C, corresponding to \( Re = 1090 \). The free-stream velocity over the permeable cylinder was 1.19
m/s, corresponding to \( Re = 1070 \); the flow conditions are similar within 2%.
FIGURE 5.15: Velocity profiles in the wake behind solid and permeable cylinders.
These profiles can be used in conjunction with Eq. (5.37) to calculate a drag coefficient of 1.16 for the solid cylinder and 1.65 for the permeable cylinder. The drag on the solid cylinder is slightly higher than expected (other data show $c_d = 1.0$), but an increase in drag on the permeable cylinder was observed under the same flow conditions. In fact, a 42% increase was observed at a permeability ratio of $6.6 \times 10^{-5}$; this data point is superimposed on Fig. 5.13. At this value of permeability ratio, the computational model predicts a drag ratio of 1.25; the experimental results differ by only 14%. Based on this piece of data, the computational results appear to be realistic.

It is possible, as Masliyah and Polikar [1980] suggested, that the increase in drag observed for the permeable body could be due to surface roughness. However, arguments can also be made that the surface roughness might decrease the drag, in which case the drag increase must be a result of the permeability of the body. Schlichting [1979] discusses the effect of roughness on drag in the context of the boundary layer on the cylinder, in which the transition to turbulence in the boundary layer can be promoted with roughness elements beyond a certain size. A turbulent boundary layer is less susceptible to separation on the curve of the cylinder, so if the drag on the cylinder is predominantly caused by pressure effects, as opposed to skin-friction effects, then delaying separation will reduce the drag coefficient. At $Re = 1000$ as in these experiments, the pressure drag is nearly dominant, while for $Re \leq 100$, the skin-friction drag is appreciable. As a result, Masliyah and Polikar's conjecture may be applicable to their results, but not necessarily to these.

5.3.5 Lift and drag on a permeable cylinder near a wall

Using the computational domain shown in Fig. 5.5 and the techniques described earlier, the forces on a permeable cylinder moving relative to a wall in a quiescent fluid were studied. There are three independent dimensionless parameters in this study: the Reynolds number and permeability ratio, as before, and the "gap ratio", or the ratio of the distance from the cylinder (as defined in Fig. 5.3) to the wall divided by the cylinder diameter. The gap ratio is represented by the symbol $\gamma^* = y/D_{cl}$. For brevity, a parametric study was run to vary for only a select number of conditions. Figures 5.16-5.21 present the lift coefficients and drag ratios for this geometry as a function of permeability ratio, for $Re = 10, 100$ and 1000 and $\gamma^* = 0.10, 0.01$ and 0.001. These values were chosen as representative values found in CFBs, and lift and drag were computed for permeability ratios between $10^{-5}$ and $10^{-3}$ for the same reason.
FIGURE 5.16: Drag ratio vs. permeability ratio, $Re = 10$

FIGURE 5.17: Lift coefficient vs. permeability ratio, $Re = 10$
FIGURE 5.18: Drag ratio vs. permeability ratio, $Re = 100$

FIGURE 5.19: Lift coefficient vs. permeability ratio, $Re = 100$
FIGURE 5.20: Drag ratio vs. permeability ratio, $Re = 1000$

FIGURE 5.21: Lift coefficient vs. permeability ratio, $Re = 1000$
These results show that the drag increases by a factor of two to three, increasing with proximity to the wall. Similarly, there is an appreciable lift coefficient that also increases as the gap ratio decreases. Depending on the speed, gap size and permeability, the lift coefficient is roughly 0.5-1.5. Both lift coefficient and drag ratio approach zero, as expected, as \( y^* \) increases. Interestingly enough, there is little change in either lift or drag from \( y^* = 0.01 \) to \( y^* = 0.001 \). This is likely due to the fact that such a small gap creates a flow resistance of similar magnitude to the permeable body itself. As the gap size decreases, the flow resistance in the gap may increase but that region becomes negligible relative to the permeable cylindrical region and therefore has a negligible effect on the overall results. The presence of this lifting force also helps explain the existence of the contact gap between the cluster and the wall which was measured by Lints and Glicksman [1993].

5.3.6 Application of model to experimental observations
With some understanding of the physical behavior of a permeable cylinder moving near a wall, what remains to be seen is if this behavior is consistent with the experimental results obtained for clusters in a CFB. Specifically, do the computed drag coefficients help explain the measured cluster velocities and do the computed lift coefficients help explain the measured cluster-wall contact times.

5.3.6.1 Computed drag vs. measured terminal velocities
As discussed in conjunction with Eq. (5.1), simple aerodynamic drag arguments over-predict the terminal velocity of a cluster of sand particles by a factor of about three and these arguments over-predict the terminal velocity of a cluster of steel particles by a factor of roughly five. This indicates that the aerodynamic drag analysis under-predicts the actual drag by a factor of roughly 10-25. Given the results from the computational model, it appears that the combination of permeability and wall proximity only double or triple the drag relative to the aerodynamic drag. It should be noted, though, that the many simplifications in this model can contribute to this shortfall. For example, neglecting the gas flow opposing the cluster, strictly speaking, is not correct. Given typical cluster sizes and distances from the wall, the gas velocity in the boundary layer might be 10-50% of the gas superficial velocity in the CFB. This represents a substantial velocity opposing the motion of the cluster and would create a significant increase in the drag force on a cluster. However, the applicability of single-phase boundary layer calculations to gas-particle flows are suspect, and the gas flow in the wall region is expected to be lower in a CFB. Also, the shape of the cluster is not necessarily regular. Analysis by Lim, et al. [1996] suggests that considering an elliptical shape for a cluster can double the drag coefficient in unbounded flow relative to a round body, while Potter and Foss [1982] present similar results for objects of square cross-section. Simply applied to the current near-wall drag results, this would suggest a four- or six-fold increase in drag which begins

124
to approach the increase required to explain the observed velocities. The fact that there is a net drag force on the cluster of particles also implies that a cluster will deform and not be round or even have a constant cross-sectional shape. The effect of shape factor is also likely to be different when considering the presence of the wall, however, and this may increase the drag ratios even further.

5.3.6.2 Computed lift vs. measured contact times
The laws of physics can be used to apply the lift and drag coefficients to understand the cluster-wall contact interaction. Considering force balances on the cluster in both the vertical \( (x) \) and lateral \( (y) \) directions, the following differential equations can be derived to describe the motion of a cylindrical cluster:

\[
x'' = g - 2 \frac{c_d \rho f (x')^2}{\pi \rho_s (1 - \epsilon_{cl}) D_{cl}}
\]

\[
y'' = 2 \frac{c_l \rho f (x')^2}{\pi \rho_s (1 - \epsilon_{cl}) D_{cl}}
\]  

where the superscripts ' and " represent the first and second derivatives with respect to time. In both Eqs. (5.38) and (5.39), added mass terms have been neglected. Solutions of these equations for \( x(t) \) and \( y(t) \) are clearly difficult – the equations are coupled, and they are both non-linear with non-constant coefficients (both \( c_d \) and \( c_l \) vary with \( x' \) and \( y \)). With a few simple assumptions, however, a solution can be generated for comparison to the CFB experiments.

First, consider that the cluster velocity is nearly constant as a function of time, at a value of 1 m/s perhaps. This results in a constant value for the lift coefficient as well, say about 0.5 judging from the results presented in Fig. 5.19. Assuming the other parameters to be constant and assuming initial values \( y(0) = y'(0) = 0 \), Eq. (5.39) can be integrated to yield:

\[
y(t) = \frac{1}{\pi \rho_s (1 - \epsilon_{cl}) D_{cl}}
\]

Furthermore, it can be argued that at some critical distance \( y_{cr} \) from the wall, the cluster will either be far enough from the wall to have a negligible contribution to heat transfer, or it will stray into a region of high enough gas velocity to be dragged upward with the core flow. Based on heat transfer arguments, a cluster-wall gap of 5 mm results in a thermal resistance similar in magnitude to that from gas convection in a CFB; it can also be shown that 5 mm is about the thickness of the
inner layer of the turbulent boundary layer. Taking $y_{cr} = 5$ mm, then, a solution for the time to reach $y_{cr}$ can be obtained from Eq. (5.40):

$$t = \sqrt{\frac{y_{cr}}{c_l u_{cl}^2} \frac{\rho_* (1 - \varepsilon_{cl}) \pi D_{cl}}{c_l u_{cl}^2}}$$

(5.41)

Values for the other parameters can be estimated: $\rho_* = 6000$ as with steel particles, $\varepsilon_{cl} = 90\%$, $D_{cl} = 1$ cm, $c_l = 0.5$. Substituting these values into Eq. (5.41) yields $t = 0.4$ seconds, which is about the same as the average measurements for steel. To be able to predict the contact time within an order of magnitude using such a simple analysis is excellent, especially considering the uncertainty of the values for $\varepsilon_{cl}$ and $D_{cl}$ as well as the approximations made to linearize the differential equation. The most important point that can be drawn from Eq. (5.41) is this: the contact time predicted varies with $\sqrt{\rho_*}$, which is precisely the trend observed in the experimental data taken in the CFB. For a factor of 2.6 variation in $\rho_*$, a factor of about 1.8 variation in $t$ was observed; Eq. (5.41) would predict a factor of 1.6 variation in $t$ for the same variation in $\rho_*$. This agreement is extremely close, again considering the variability in the other parameters, and it lends a substantial amount of credence to the hypothesis that the lift force on the cluster governs the cluster-wall contact time. Figure 5.22 presents the magnitudes and trends for contact times predicted by Eq. (5.41) superimposed upon the experimental results.

![Graph showing cluster-wall contact time vs. solid-gas density ratio](image)

**FIGURE 5.22:** Application of lift-model predictions to experimental results
This analysis can be further refined by integrating Eq. (5.39) while accounting for the variation in lift coefficient with distance from the wall. Figure 5.23 presents the data and curve fit for lift coefficient as a function of distance for a 1 cm cylinder with a 10% solid fraction moving at 1 m/s in room temperature air. The functional form of the curve fit is \( c_l = 0.35 - 0.29 \log(y/D_c) \).

![Graph](image)

**FIGURE 5.23: Lift coefficient vs. distance from wall**

Since lift varies with the square of velocity, it can be assumed most of the lifting force occurs at the maximum velocity so that the \((x')^2\) term in Eq. (5.39) is constant. With this and the functional form of \( c_l \), Eq. (5.39) can be numerically integrated to yield a similar result as before – it takes 0.36 seconds for the cylinder to travel 5 mm laterally. This level of analysis retains the dependence of contact time on solid density observed in Fig. 5.22.
5.4 SUMMARY OF MODELING RESULTS

Analytical and computational modeling was performed in order to understand the physical phenomena governing the motion of clusters near the wall of a circulating fluidized bed. The analytical model for drag on a cluster and terminal velocity of a cluster agrees very well with current and previous experimental observations (see Fig. 5.2), but the model itself ignores the presence of the wall. Computational results show that the wall can have a significant effect on drag, although the computational results only give fair agreement in the prediction of either drag or terminal velocity. However, the computational model, when applied to analyze a lifting force as the mechanism for cluster shedding, gives reasonable agreement of both the magnitudes and trends observed for cluster-wall contact times. Several yet-unmeasured parameters, such as the exact size and shape of clusters, enter the cluster-motion model of Eqs. (5.38) and (5.39) and may very well contribute to the discrepancies between the computational results and the experimental observations.

A few potential shortcomings of the computational model need to be noted. One is that the effect of turbulence has been ignored. It was shown that reasonable agreement could be achieved between the computational model and experimental results for unbounded flow past solid and permeable cylinders. However, the presence of the wall complicates the flow geometry and wake structure, adding some amount of uncertainty to the results obtained for those cases. Another is that the state of knowledge of the physics of flow in porous media indicates that D'Arcy's Law does not hold for high porosities and high fluid velocities. This effect was first analyzed by Brinkman [1947] and Brinkman's momentum equation was used in place of D'Arcy's Law in the computations for unbounded flow past permeable spheres performed by Nandakumar and Masliyah [1982]. However, closer inspection of Brinkman's work shows that the Carman-Kozeny relation — presented as Eq. (5.34) — over-predicts permeability for solid fractions less than 10%. Given typical values of solid fractions for clusters, this is not expected to have a significant effect on the results. Finally, there is the fact that the clusters are not rigid, permeable bodies — they are composed of unconnected particles. As a result, the lift and drag forces will result in some deformation of the clusters and the actual forces will deviate somewhat because of this. However, the computed results at least show the presence of these forces and their dependence on the permeability of the cluster.
5.5 REFERENCES


S. Ergun, Fluid flow through packed columns, Chemical Engineering Progress, 48, 89 (1952).


6. CONCLUSIONS AND RECOMMENDATIONS

6.1 SUMMARY OF RESULTS AND CONCLUSIONS

6.1.1 Motivation
Circulating fluidized beds have been gaining acceptance as environmentally-sound systems for coal combustion, in which limestone particles are used to absorb gaseous sulfur emissions. Convective heat transfer at the wall of a circulating fluidized bed (CFB) can be modeled as having two parallel contributions, gas convection and particle convection. The particle-convective contribution to heat transfer depends primarily on the near-wall hydrodynamics of clusters of solid particles, so that a hydrodynamically similar cold-flow model can be used to study particle convection.

6.1.2 Experimental results and conclusions

6.1.2.1 Experimental methods
A new technique has been developed to measure some of the hydrodynamic characteristics of the flow of clusters of solid particles near the wall of a CFB. This technique is called Thermal Image Velocimetry (TIV), and it employs a thermal-tracer technique, a wall that is transparent to infrared radiation and an infrared camera to visualize the flow near the wall. Measurements of cluster velocities and residence times of clusters near the wall, and qualitative measurements of fractional wall coverage by clusters, are possible with the TIV technique.

6.1.2.2 Cluster-wall contact measurements

On a flat wall
Cluster-wall contact times between about 0.2 and 0.5 seconds were measured, with increases in contact times seen with increasing density of the solid particles. These contact times can be used to predict heat-transfer rates consistent with those observed in CFBs. The trend of shorter contact times with lower solid densities may be indicative of higher heat transfer rates in pressurized CFBs, since the solid-gas density ratio is lower in those systems.
**On a membrane wall**
The average cluster-wall contact time appears to be lower on a membrane wall compared to a flat wall, as does the fraction of the wall covered by clusters. Although the shorter contact time increases the particle-convective contribution to heat transfer, the lower wall coverage indicates less surface area on which particle convection plays a role. These results can be used to predict a net reduction in convective heat transfer, which are consistent with observations in other systems. However, some unavoidable wall roughness in the modified test section in these experiments degrade confidence in the observed results.

**Probabilistic distribution of contact times**
The contact-time measurements on both the flat and membrane walls were found to follow a gamma probability density function, which indicates that some random motion might govern the deposition and/or shedding of clusters at the wall. The probabilistic behavior can also be used to quantify differences in particle convection between thermally developing and developed regions.

### 6.1.2.3 Cluster velocity measurements
The clusters near the wall appear to accelerate as if in gravitational free-fall, approaching a terminal velocity slightly higher than 1 m/s. This terminal velocity is roughly independent of operating conditions, and is consistent with velocities measured in other CFBs. The acceleration of clusters cannot be ignored when calculating heat transfer based on the particle-convection model, especially if the average contact time is short; ignoring this effect can result in the over-prediction of the cluster contribution to heat-transfer rates by about 50%.

### 6.1.2 Modeling results and conclusions
#### 6.1.2.1 Analytical modeling
**Cluster-wall contact times**
Assuming that the cluster-wall contact time varies with the time-scales for turbulent motion in single-phase flow, a relationship between the characteristic time-scale and the CFB operating conditions can be derived. This analysis shows a theoretical dependence on solid density, as observed experimentally, but it also predicts a similar dependence on gas velocity, for which no dependence was observed. This indicates that the eddy motion may not govern the cluster-wall contact, although the level of error in the experiments may be obscuring an otherwise unseen dependence on gas velocity.
Cluster descent velocities
The descent velocities of clusters in a CFB, observed here and elsewhere, are significantly lower than the terminal velocities that would be predicted by a simple aerodynamic analysis. Assuming that additional drag results from air passing through a cluster of particles, it can be shown that the terminal velocity of a cluster is a function of the Archimedes number. This analysis applies fairly well to a large number of measurements. However, a few measurements deviate significantly, and this analysis ignores the effect of the wall on the flow field in the vicinity of the cluster.

6.1.2.2 Computational modeling
A computational model was employed in order to incorporate all of the physically relevant phenomena affecting the flow of a cluster near a wall, including the permeability of a cluster and the proximity to the wall. Such a model was expected to unify the predictions for both cluster velocities and contact times by identifying the axial and lateral forces acting on a cluster resulting from the same physical phenomena. This modeling method shows that the drag on a porous body far from the wall can be around 50% higher than on a similar solid body. Furthermore, the drag on a porous body is roughly twice that on an unbounded solid cylinder, although not sufficiently higher to predict the magnitudes of cluster velocities observed. There are other factors that may contribute to higher drag, such as the actual shape, size and porosity of a cluster; these factors are more difficult to model, however. The computational model also predicts significant lateral, or lifting, forces on a cluster near a wall, indicating that clusters are shed from the wall in a gradual manner. The contact times predicted by the lifting theory are about 0.2-0.4 s and are in fairly good agreement with both the magnitudes measured and the trends observed with variations in the density of the solid particles.

6.1.3 Implications for CFB design
Two significant conclusions that effect the design for heat transfer in CFBs emerge from this work. One conclusion is that lighter (smaller or less dense) particles tend to have shorter residence times at the wall, which implies that a CFB using lighter particles should experience higher bed-to-wall convective heat transfer. The particles in CFBs that run at elevated pressures are less dense relative to the gas phase, and as discussed previously, there have been reports of higher heat transfer rates with increasing system pressure in CFBs. Given that cluster-wall contact times appear to be governed by the flow around the clusters, the inertia of the gas phase should play an important role in this aspect of convective heat transfer. The other conclusion is that higher convective heat transfer coefficients are expected, on average, for flat walls as compared to finned-tube or membrane walls. Although measurements reported here indicate that the contribution from particle convection will be greater on membrane walls, less of the wall surface is actually covered
by clusters. The latter effect takes away more from convective heat transfer than the particle convection adds, although these results may vary from system to system. A more detailed analysis of the finned-tube configuration, accounting for the variation of heat transfer coefficients on the different surfaces, should be performed for a particular system configuration to obtain a more accurate prediction of the effect of wall geometry on heat transfer.
6.2 RECOMMENDATIONS FOR FUTURE WORK

6.2.1 Experimental methods
The TIV technique that was designed for this project has proven to be effective, but it is really a "first generation" system and parts of it are still in the developmental stages. For example, the method for data reduction (reviewing videotapes) is manual and therefore very time-consuming. Automating this process would require additional hardware and software, at a moderate expense, as well as more rigorous criteria for defining clusters in a given region. Other refinements could include updating the infrared imaging technology (newer cameras have better time and temperature resolution), building a stronger wall in the test section out of a typical "infrared window" material (such as zinc sulfide, zinc selenide or germanium), or designing a better test section to accommodate the membrane-wall configuration. These refinements are quite expensive, however, which is why they were not pursued for this project. However, automation or refinement would cut down on experimental error which in turn would instill more confidence in the experimental results.

6.2.2 Computational methods
Some simplifications were made in using the CFD model to investigate the forces on a cluster near a wall. The most prominent of these was ignoring turbulence in the analysis. Comparisons to other computations and experiments showed that this was sufficient when neglecting the presence of the wall, but it is unclear if the same assumption can or should be applied to the near-wall analysis. As discussed earlier, the effects of turbulence were ignored for efficiency. Another simplification to the CFD model was the application of D'Arcy's Law to describe the flow within the cluster. It is unclear whether or not D'Arcy's Law applies at relatively low solid fractions, and even if so, it is unclear if the distribution of particles throughout the cluster is sufficiently uniform to allow the use of a single value of permeability. A more accurate model would consider the cluster region to contain individual solid particles, rather than a permeable continuum; however, such a modeling effort would be a Ph.D. thesis unto itself. There is also, of course, the issue of what shape is the proper one to use when modeling a cluster. That question can only be answered by more in-depth investigation in CFBs, and even then, the answers probably vary from system to system. Finally, as with any computational modeling, experimental substantiation is always helpful. More wind tunnel testing on permeable bodies near walls would certainly put to rest some of the questions that still remain.
6.2.3 Long-term impact on heat transfer correlations

The convective heat transfer model of Eq. (2.8) contains several hydrodynamic parameters, one of which has been explored in detail in this study. Through experimentation and modeling, mechanisms for cluster-wall contact have been revealed and can therefore be generalized to apply to a variety of similar systems. Unfortunately, the state of basic knowledge is low for the other hydrodynamic parameters governing heat transfer; there are some empirical correlations, but no physical mechanisms have been identified to explain certain levels of wall coverage, or certain cluster sizes or cluster solid fractions. Applying the approach taken here for obtaining basic physical knowledge to the other hydrodynamic parameters can transform Eq. (2.8) into a model that can be used to calculate heat transfer based solely on operating conditions. Current CFB heat exchanger designs typically rely on empirical $h$ vs. $\varepsilon_{ov}$ correlations, which only apply to the specific configurations for which the correlations were obtained. Better understanding of the physics in the wall layer, similar to understanding the physics in a boundary layer in single-phase flow, is a key to more useful and accurate analysis of heat transfer at the wall of a CFB.
A. Summary of Particle Characteristics

Three sets of particles were used for most of the experiments in this study: one grade of steel particles and two grades of sand. The following table and charts summarize the characteristics of these particles that are relevant to CFBs: density, average diameter, sphericity, loose-packed void fraction, and distribution of particle sizes. The measurement methods are described in Chapter 3.

<table>
<thead>
<tr>
<th>material</th>
<th>mean size</th>
<th>density</th>
<th>average sphericity</th>
<th>loose-packed void fraction</th>
<th>calculated min. fl. vel.</th>
</tr>
</thead>
<tbody>
<tr>
<td>steel</td>
<td>69 µm</td>
<td>6980 kg/m³</td>
<td>0.73±0.13</td>
<td>0.65±0.02</td>
<td>1.22 cm/s</td>
</tr>
<tr>
<td>sand</td>
<td>128 µm</td>
<td>2660 kg/m³</td>
<td>0.74±0.06</td>
<td>0.44±0.01</td>
<td>1.60 cm/s</td>
</tr>
<tr>
<td>sand</td>
<td>182 µm</td>
<td>2650 kg/m³</td>
<td>0.79±0.05</td>
<td>0.43±0.01</td>
<td>3.21 cm/s</td>
</tr>
</tbody>
</table>

TABLE A.1: Particle characteristics

![Graph](image.png)

FIGURE A.1: Particle-size distribution for steel particles
FIGURE A.2: Particle-size distribution for 128 μm sand particles

FIGURE A.3: Particle-size distribution for 182 μm sand particles
B. Design Data for Pressure Transducers

The CFB riser has eleven pressure taps along its length, allowing for ten differential pressure measurements. These measurements allow for the calculation of the average solid concentration as a function of riser height, as described in Chapter 3. The following table contains design information pertinent to the installation of the pressure transducers: the transducers are numbered from the bottom of the riser; the rated range of the transducer is that quoted by the manufacturer; the high/low tap locations are those corresponding to the high and low pressure readings; the dimensionless location is the fraction of riser height corresponding to the midpoint between two taps measured from the air distributor; and the calibration function converts the voltage signal from the transducer into pressure expressed in inches of water column.

<table>
<thead>
<tr>
<th>transducer number</th>
<th>transducer rated range</th>
<th>high/low tap location</th>
<th>dimensionless location</th>
<th>calibration function</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1240 Pa</td>
<td>1.9/10.4 cm</td>
<td>0.025</td>
<td>$\Delta P = 1.236 , V - 1.356$</td>
</tr>
<tr>
<td>2</td>
<td>1240 Pa</td>
<td>10.4/17.3 cm</td>
<td>0.057</td>
<td>$\Delta P = 1.233 , V - 1.351$</td>
</tr>
<tr>
<td>3</td>
<td>1240 Pa</td>
<td>17.3/34.9 cm</td>
<td>0.11</td>
<td>$\Delta P = 1.220 , V - 1.332$</td>
</tr>
<tr>
<td>4</td>
<td>500 Pa</td>
<td>34.9/54.9 cm</td>
<td>0.18</td>
<td>$\Delta P = 0.4815 , V - 0.6171$</td>
</tr>
<tr>
<td>5</td>
<td>500 Pa</td>
<td>54.9/80.6 cm</td>
<td>0.28</td>
<td>$\Delta P = 0.4799 , V - 0.5769$</td>
</tr>
<tr>
<td>6</td>
<td>500 Pa</td>
<td>80.6/108 cm</td>
<td>0.39</td>
<td>$\Delta P = 0.4779 , V - 0.5328$</td>
</tr>
<tr>
<td>7</td>
<td>500 Pa</td>
<td>108/138 cm</td>
<td>0.50</td>
<td>$\Delta P = 0.4818 , V - 0.6174$</td>
</tr>
<tr>
<td>8</td>
<td>500 Pa</td>
<td>138/168 cm</td>
<td>0.63</td>
<td>$\Delta P = 0.4751 , V - 0.5448$</td>
</tr>
<tr>
<td>9</td>
<td>500 Pa</td>
<td>168/199 cm</td>
<td>0.75</td>
<td>$\Delta P = 0.4821 , V - 0.6005$</td>
</tr>
<tr>
<td>10</td>
<td>500 Pa</td>
<td>199/229 cm</td>
<td>0.88</td>
<td>$\Delta P = 0.4764 , V - 0.6001$</td>
</tr>
</tbody>
</table>

TABLE B.1: Pressure tap and transducer characteristics
C. Probability analysis for data reduction

A simple example can illustrate the derivation of Eq. (3.8) and subsequent probabilistic results. Consider that cluster contact times are distributed according to the discrete function shown in Figure C.1, such that 10 clusters travel for 1 second, 20 travel for 2 seconds, etc.

![Figure C.1: Graphical example of $p_d(t)$](image)

By assuming that clusters are heated at any fraction of their travel time and therefore result in the measurement of some lesser or equal contact time, a set of measurements as in Figure C.2 would be expected.

![Figure C.2: Graphical example of $p_d(t)$](image)
In other words, the 10 clusters that travel for 5 seconds have an equal probability of being heated such that the measured travel time is either 1, 2, 3, 4 or 5 seconds. As a result, one-fifth of the total number of clusters – or two clusters, since there are 10 total – will be measured in each time slot less than or equal to 5 seconds. Similarly, one-fourth of the total number of clusters traveling for 4 seconds can be measured as having traveled some time less than or equal to 4 seconds. Relating this to the continuous domain, the number of clusters traveling for a given time is \( p_c(t) \), the fraction of those measured at some lesser time in the interval \( dt \) is \( dt/\tau \); therefore, the total number of measurements appearing for a given value of \( \tau \) is the summation of \( \{ p_c(t)/\tau \} dt \) for all \( t \geq \tau \). In the continuous limit, this can be expressed as:

\[
p_s(\tau) = \int_{\tau}^{t_{\text{max}}} \frac{p_c(t)}{t} \, dt
\]

(C.1)

This is also given as Eq. (3.8).

It can also be shown that \( t_{\text{avg}} = 2 \tau_{\text{avg}} \) for any distribution \( p_c(t) \) or \( p_s(\tau) \), as was concluded following Eq. (3.11). Given the definition of \( \tau_{\text{avg}} \):

\[
\tau_{\text{avg}} = \int_{0}^{t_{\text{max}}} \tau \, p_s(\tau) \, d\tau
\]

(C.2)

integrating by parts yields:

\[
\tau_{\text{avg}} = \left[ \frac{\tau^2}{2} p_s(\tau) \right]_{0}^{t_{\text{max}}} - \int_{0}^{t_{\text{max}}} \frac{\tau^2}{2} \frac{dp_s}{d\tau} \, d\tau
\]

(C.3)

The first term on the right-hand side is zero, since \( \tau^2/2 = 0 \) for \( \tau = 0 \) and \( p_s(\tau) = 0 \) for \( \tau = t_{\text{max}} \) using the definition in Eq. (C.1). For the second term, differentiating Eq. (C.1) and substituting the result into Eq. (C.3) yields:

\[
\tau_{\text{avg}} = \int_{0}^{t_{\text{max}}} \frac{\tau^2}{2} \frac{p_c(\tau)}{\tau} \, d\tau = \frac{1}{2} \int_{0}^{t_{\text{max}}} \tau \, p_c(\tau) \, d\tau
\]

(C.4)

The right-hand side of Eq. (C.4) is simply the definition of \( t_{\text{avg}} \), so that \( \tau_{\text{avg}} = 1/2 \, t_{\text{avg}} \) regardless of the definition of \( p_c(t) \). This is the result stated following Eq. (3.11).
D. Calibration of the TIV system for temperature measurements

The goal of this analysis is to determine the true temperature on the surface of a cluster in the TIV test section given the temperature reading from the infrared camera. To be as general as possible, it is necessary to know, or at least estimate within a couple of degrees, the temperatures of both the surroundings (the room) and the sheet of plastic.

D.1 THE INFRARED CAMERA AND SOURCES OF RADIANT ENERGY
The infrared camera determines a temperature as follows. A detector senses the radiant energy incident upon it from a given point in space; using the laws of thermal radiation and assuming that the object emitting to it has a surface emissivity of 0.95, a temperature is calculated. In this system, however, the situation is much more complex due to the fact that the LDPE wall both attenuates, emits and reflects radiant energy. Therefore, in the current set-up, there are three sources of radiant energy incident upon the camera and its detector: energy emitted from a cluster behind the plastic, less any energy reflected backward and attenuated; energy emitted by the LDPE itself; and energy emitted by the surroundings that is reflected off the front of the LDPE. In this analysis, it is assumed that the cluster blocks any radiation from the background in the CFB; this assumption is justified later. Figure D.1 gives a schematic of the energy sources and their interactions with both the plastic and the camera.

All of the materials are assumed to behave as ideal gray-bodies in the operating range of the camera; in other words, the radiant properties of their surfaces (emissivity – ε, reflectivity – ρ, and transmissivity – τ) are constant, and ε + τ + ρ = 1 for a given material. This limits the analysis to being valid only for wavelengths to which the infrared camera is sensitive, or about 8-12 μm (the gray-body assumption is justified later in the analysis). From the camera’s perspective as in Fig. D.1, a temperature will be determined from the following energy balance:

\[ 0.95 \sigma T_{cam}^4 = \varepsilon_{cl} \tau_{pl} (1 - \rho_{pl}) \sigma T_{cl}^4 + \varepsilon_{pl} \sigma T_{pl}^4 + \rho_{pl} \sigma T_{rm}^4 \]  

(D.1)

From this, the functional form for the true temperature of the cluster can be found:

\[ T_{cl}^4 = \frac{0.95 T_{cam}^4 - \varepsilon_{pl} T_{pl}^4 - \rho_{pl} T_{rm}^4}{\varepsilon_{cl} \tau_{pl} (1 - \rho_{pl})} \]  

(D.2)
Using Eq. (D.2), the system can be calibrated by obtaining measurements of the emissivity of a cluster of particles and the LDPE sheet, as well as of the transmissivity and reflectivity of the LDPE.

FIGURE D.1: Radiant energy from the CFB system incident upon the IR camera
D.2 MEASUREMENT OF SURFACE PROPERTIES

D.2.1 Cluster emissivity and reflectivity

The particles used in this calibration experiments are the 69 μm steel particles, the ones used in the actual temperature measurements reported in Chapter 3. To estimate the emissivity of a cluster of steel particles, the temperature on the surface of a pile of steel powder was measured with both a thermocouple and the infrared camera. Within 0.5°C (the resolution of the thermocouple), both methods yielded the same result: 27°C from the thermocouple, 26.5°C from the IR camera. Since the camera assumes a surface emissivity of 0.95, and since the camera appears to yield the correct temperature, the emissivity of the surface of the cluster can be taken to be ε_c = 0.95. Passing a warm object (e.g., a human hand at about 36°C) in front of the steel pile but not directly in front of the camera causes no change in the camera reading, indicating no reflection of the energy from the warmer object. Therefore, the reflectivity of the surface of the cluster can be taken as ρ_c = 0, and this is not expected to change for a more loosely-packed cluster.

D.2.2 Cluster transmissivity

The above test for cluster emissivity was performed using a densely-packed pile of steel powder; in a CFB, though, the volumetric solid concentration in a cluster might be only 5-10% perhaps. This is a potential concern, since the background temperature in the operating CFB could be a substantial source of radiation incident on the camera. For radiation through a one-dimensional layer of particles, the transmissivity can be defined over the hemisphere above the surface as follows [Hottel & Sarofim, 1967]:

\[ \tau_c = 2 \int_0^1 z \exp \left( -\frac{\kappa \Delta x}{z} \right) dz \]  \hspace{1cm} (D.3)

where Δx is the thickness of the cluster and κ is the "extinction coefficient", given by Hottel & Sarofim [1967] for opaque convex surfaces:

\[ \kappa = \frac{2 \ c_c}{3 \ d_p} \]  \hspace{1cm} (D.4)

κ is a function of the solid concentration in the cluster (c_c) and the size of the particles (d_p); the product κΔx is also known as the "optical thickness" of the cluster. From Eqs. (D.3) and (D.4) it becomes apparent that the transmissivity of the cluster is a function of the size of the particles, their concentration in the cluster, and the thickness of the cluster. A variety of parametric studies can be run to find τ_c, but one well-chosen case can be sufficiently illustrative. For example, given d_p = 69 μm in diameter and a cluster thickness of 5 mm, Figure D.2 shows that the cluster
transmissivity is negligible for concentrations greater than 5%, which is at the low end of expected values.

![Cluster transmissivity vs. particle concentration](image)

**FIGURE D.2: Cluster transmissivity vs. particle concentration**

In fact, the thickness of the cluster has to be greatly reduced in order to transmit an appreciable amount of radiation at the concentrations expected. Therefore, the transmissivity of the cluster can be taken to be \( \tau_{cl} = 0 \); in other words, radiation from behind the cluster does not reach the camera.

**D.2.3 Gray-body assumption for LDPE**

Figure D.3 presents measurements of normal transmissivity vs. wavelength for 0.5-mm thick LDPE, the same material used in the TTV test section, as measured using an FT-IR spectrometer from Bio-Rad Inc. of Cambridge, MA. In the wavelength range from 8-12 \( \mu \text{m} \), the transmissivity is approximately constant at somewhere around 40-50%. It seems fair to assume that the other optical properties are also constant in this range, indicating that the LDPE sheet can be treated as a gray body (optical properties independent of wavelength).
D.2.4 LDPE reflectivity

To measure the reflectivity of the LDPE, a warm object (again, a human hand) was held in front of the plastic but beyond the field of view of the camera. In this way, energy from the object should reflect off the plastic and into the camera. Given that the temperature of the room and the plastic sheet are about the same (26°C) and the surface of a hand is 36°C (slightly cooler than body temperature), the camera indicates that the temperature of the plastic is 25.1°C with the hand reflecting into the camera and 24.1°C without the hand reflecting into the camera. To analyze this data and extract a value for \( \rho_{pl} \), the principles of thermal radiation outlined earlier can be used.

Without the reflection from the hand, the sources of energy incident on the infrared camera are the surroundings from behind the plastic, the surroundings reflected off the front of the plastic, and the emission from the plastic itself. In this case, an energy balance yields:

\[
0.95 \ T_{\text{cam}}^4 = \ \tau_{pl} (1 - \rho_{pl}) \ T_{rm}^4 + \ \varepsilon_{pl} \ T_{pl}^4 + \ \rho_{pl} \ T_{rm}^4 \quad (D.5)
\]

Since the temperature of the room and the temperature of the plastic are the same, and since \( 1 - \rho_{pl} = \tau_{pl} + \varepsilon_{pl} \) (the gray-body assumption), Eq. (D.5) reduces to:

\[
0.95 \ T_{\text{cam}}^4 = \left( \tau_{pl}^2 + \tau_{pl} \varepsilon_{pl} + \varepsilon_{pl} + \rho_{pl} \right) \ T_{rm}^4 = (X + \rho_{pl}) \ T_{rm}^4 \quad (D.6)
\]
where $X$ is used as shorthand for a function of the transmissivity and emissivity of the plastic. Given that $T_{rm} = T_{pl} = 26^\circ$C and that $T_{cam} = 24.1^\circ$C without the hand reflecting on the plastic, Eq. (D.6) can be used to calculate $X + \rho_{pl} = 0.926$.

With the hand in place, the reflection from the hand adds an additional source term, yielding the following energy balance:

$$0.95 \cdot T^4_{cam} = X \cdot T^4_{rm} + \rho_{pl} \cdot F_{h,p} \cdot T^4_h + \rho_{pl} \cdot (1 - F_{h,p}) \cdot T^4_{rm}$$  \hspace{1cm} (D.7)

In Eq. (D.7), $F_{h,p}$ is the radiation shape factor between the hand and the LDPE sheet. This shape factor is a function of the size of the hand, the size of the spot where the temperature is being measured by the camera, and the distance between the hand and the plastic. Approximating the geometry by concentric disks, the following formula gives the shape factor [Modest, 1993]:

$$F_{h,p} = \left( 1 + \frac{s^2 + r_h^2}{r_p^2} - \left[ \left( 1 + \frac{s^2 + r_h^2}{r_p^2} \right)^{1/2} \right]^2 \right)$$  \hspace{1cm} (D.8)

where $s$ is the spacing between the hand and the plastic, $r_h$ is the approximate diameter of the hand and $r_p$ is the diameter of the spot being measured on the plastic. Given that the size of a hand is about 10 cm, the size of the spot being measured is about 1 cm and the hand was about 10 cm from the plastic, $F_{h,p} = 0.998$, or $F_{h,p} = 1$. This indicates that there is no reflection from the room off the front of the plastic into the camera, and Eq. (D.7) reduces to:

$$0.95 T^4_{cam} = X T^4_{rm} + \rho_{pl} T^4_h$$  \hspace{1cm} (D.9)

Eqs. (D.6) and (D.9) comprise a set of equations for $X$ and $\rho_{pl}$. Given that $T_{cam} = 25.1^\circ$C with $T_{rm} = 26^\circ$C and $T_h = 36^\circ$C, the solution $\rho_{pl} = 0.09$ can be obtained. The value of $X$ and the gray-body relation could be used to determine $\tau_{pl}$ and $\varepsilon_{pl}$; however, the function $X$ is very sensitive to variations in $\tau$ and $\varepsilon$, and small variations in those properties result in large solution errors. A more direct method for measuring these properties is required.

**D.2.5 LDPE emissivity and transmissivity**

To determine $\tau_{pl}$ and $\varepsilon_{pl}$ more directly, the temperature of a pile of steel powder was varied with a heater. The temperature indicated by the camera with the plastic sheet in place was then recorded, with the heater off while the camera readings were being noted. This configuration is the same as that in a CFB represented by Eq. (D.2), except that now $T_{pl} = T_{rm} = 29^\circ$C instead of $T_{pl} = T_{bed}$. Figure D.4 presents a curve fit of the form of Eq. (D.2) to the results obtained with the infrared camera and the pile of powder.
Although the data in Fig. D.4 appear to be linear in the range of temperatures observed, the behavior is known to be quite non-linear. Therefore, fitting a curve to this data is the most general technique. With $T_{pl} = T_{rm}$, Eq. (D.2) reduces to:

$$T_{st}^4 = \frac{0.95 T_{cam}^4 - (\epsilon_{pl} + \rho_{pl}) T_{rm}^4}{\epsilon_{st} \tau_{pl} (1 - \rho_{pl})} \quad \text{(D.10)}$$

where $T_{st}$ is the temperature on the surface of the pile of steel powder. Since $\rho_{pl}$ and $\epsilon_{st}$ are known, only two curve-fit parameters are required as Eq. (D.10) reduces even further to:

$$T_{st}^4 = A T_{cam}^4 - B T_{rm}^4 \quad \text{(D.11)}$$

The best-fit parameters are found to be $A = 2.55$ and $B = 1.50$. Comparing Eqs. (D.11) and (D.10), the values $\tau_{pl} = 0.43$ and $\epsilon_{pl} = 0.498$ can be calculated. Note that the value for transmissivity is about that expected from Fig. D.3.
D.3 CALIBRATION SUMMARY

Using the calculated values for the surface properties in the original equation governing the cluster/plastic/camera interaction, the calibration curve for the infrared camera is given by:

\[ T_{cl}^4 = 2.55 T_{camera}^4 - 1.29 T_{plastic}^4 - 0.24 T_{room}^4 \]  \hspace{1cm} (D.12)

An in situ validation of Eq. (D.12), without using the wall heater in the TIV test section, shows its accuracy. Running the CFB at a typical operating condition, the fluidization air is at a temperature of 51°C, the room at 26°C and the plastic at 33°C (all measured by thermometers or thermocouples). Under these conditions, the camera reads 38°C for the temperature of material at the wall. Using Eq. (D.12), a cluster temperature of 48°C can be calculated, which is about that expected. The temperature of the particles at the wall ought to be close to the fluidization air temperature, but slightly cooler since there is some time constant associated with bringing them to the bed temperature from the circulation loop and since they are being cooled somewhat by the ambient room condition.

D.4 REFERENCES


E. Input files for computational simulations

The input files required to run PHOENICS are listed in the following order: unbounded flow past a solid cylinder, unbounded flow past a permeable cylinder, and relative motion of a permeable cylinder to a wall in a quiescent fluid. The input files are in a form known as "PHOENICS Input Language" or "PIL". The structure and syntax of PIL can be found in the PHOENICS User's Manual (see the Concentration, Heat and Momentum, Ltd., 1991, reference at the end of Chapter 5).
** PHOENICS input file for flow past a solid cylinder, unbounded

TALK=T;RUN(1,1);VDU=V11-TERM

GROUP 1. Run title and other preliminaries
TEXT:FLOW OVER A SOLID CYLINDER IN UNBOUNDED FLOW
TITLE

** declaration of variables

REAL(PL,TUPI,VINF,DCYL,RCYL);PI=2*ASIN(1.0);TUPI=2*PI
REAL(REL,AA,DY,DYFR);INTEGER(NXF,NYP)
INTEGER(JJ1,NCYL)
INTEGER(DX1);DX1=NX/2

GROUP 3. X-direction grid specification

** polar coordinates, 120 evenly-spaced circumferential divisions

CARTES=F;
NX=120;NXF=NX/4
GRDPWR(X,NX,TUPI,1.0)

GROUP 4. Y-direction grid specification

** setting cylinder size and domain size to 48°D
** setting routine for increasing coarseness away from cylinder

RCYL=0.005;DCYL=2*RCYL
NCYL=48
YVLAST=NCYL*RCYL
DY=RCYL/50
AA=1.2905
DYFR=DY/YVLAST
NYP=5;NY=27+NYP

YFRAC(JJ)=RCYL/YVLAST/NYP
DO JJ=2,NYP
+ YFRAC(JJ)=JJ*YFRAC(1)
ENDDO
DO JJ=NYP+1,NY-2
+ JJ=JJ-1;YFRAC(JJ)=YFRAC(JJ)+DYFR;DY=DY*AA;DYFR=DY/YVLAST
ENDDO
YFRAC(NY-1)=0.95
YFRAC(NY)=1.0

GROUP 7. Variables stored, solved & named
SOLVE(P1,U1,V1)

GROUP 9. Properties of the medium (or media)

** setting flow properties - density, viscosity, velocity

REL=100.
GROUP 11. Porosities

** define a the inner cylinder to be solid

CONPOR(CYLINDER, 0.00, CELL, 1, NX, 1, NYP, 1, 1)

GROUP 13. Boundary conditions and special sources

** boundary conditions on the domain:
- ambient pressure on the outer boundary
- inflow at specified velocity from the left half
- a solid boundary surrounding the inner cylinder
- outflow at a solved velocity on the right half

PATCH(PINF, NORTH, 1, NX, NY, NY, 1, 1, 1, 1)
COVAL(PINF, P1, 1.0E+05, 0.0)
PATCH(PINF, CELL, 1, NX, NY, NY, 1, NZ, 1, 1)
COVAL(PINF, P1, 1.0E5, 0.0)
COVAL(PINF, U1, ONLYMS, 0.0); COVAL(PINF, V1, ONLYMS, 0.0)

PATCH(UPOL, CELL, 1, NXF, NY, NY, 1, NZ, 1, 1)
COVAL(UPOL, U1, FIXVAL, GRND1)
PATCH(UPOL2, CELL, 3*NXF+1, NX, NY, NY, 1, NZ, 1, 1)
COVAL(UPOL2, U1, FIXVAL, GRND1)
PATCH(VPOL, CELL, 1, NXF, NY-1, NY-1, 1, NZ, 1, 1)
COVAL(VPOL, V1, FIXVAL, GRND1)
PATCH(VPOL2, CELL, 3*NXF+1, NX, NY-1, NY-1, 1, NZ, 1, 1)
COVAL(VPOL2, V1, FIXVAL, GRND1)

WALL (WALL, NORTH, 1, NX, NYP, NYP, 1, NZ, 1, 1)

PATCH(OUTFLOW, CELL, NXF+1, 3*NXF, NY, NY, 1, NZ, 1, 1)
COVAL(OUTFLOW, U1, ONLYMS, SAME); COVAL(OUTFLOW, V1, ONLYMS, SAME)

GROUP 15. Termination of sweeps
LSWEEP = 5000

GROUP 16. Termination of iterations
LITER(P1) = 12; LITER(U1) = 6; LITER(V1) = 6

GROUP 17. Under-relaxation devices
RELAX(U1, FALSEDT, 1.0); RELAX(V1, FALSEDT, 1.0)

GROUP 19. Special calls from EARTH to GROUND
POLRA = -VINF

GROUP 20. Preliminary print-out
ECHO = T

GROUP 22. Spot-value print-out
NPLT=2;IXMON=NXF;IYMON=NYP+1

GROUP 23. Field print-out and plot control
NXPRIN=1;NYPRIN=1
PATCH(MAP,CONTR,1,NX,1,NY,1,1,1,1);PLOT(MAP,P1,1.0,10.0)
PLOT(MAP,U1,1.0,10.0);PLOT(MAP,V1,1.0,10.0)

tstswp=1

stop
*** PHOENICS input file for flow past a permeable cylinder, unbounded

TALK=T;RUN( 1, 1);VDU=X11-TERM
   GROUP 1. Run title and other preliminaries
TEXT(FLOW OVER A POROUS CYLINDER - RE = 100
TITLE

** declaration of variables

REAL(PI,TUPI,VINF,DCYL,RCYL);PI=2*ASIN(1.0);TUPI=2*PI
REAL(REL,AA,DY,DYFR,TKEIN,EPSIN);INTEGER(NXF,NYP)
REAL(DARCI)
INTEGER(JJ1,NCYL)
INTEGER(IX1);IX1=NX/2

GROUP 3. X-direction grid specification

** polar coordinates, 120 evenly-spaced circumferential divisions

CARTES=F;
NX=120;NXF=NX/4
GRDPWR(X,NX,TUPI,1.0)

GROUP 4. Y-direction grid specification

** setting cylinder size and domain size to 48*D
** setting routine for increasing coarseness away from cylinder

RCYL=0.005;DCYL=2*RCYL
NCYL=48
YVLAST=NCYL*RCYL
DY=RCYL/20
AA=1.2905
DYFR=DY/YVLAST
NYP=25;NY=23+NYP

YFRAC(1)=RCYL/YVLAST/NYP
DO JJ=2,NYP
  + YFRAC(JJ)=JJ*YFRAC(1)
ENDDO
DO JJ=NYP+1,NY-2
  + JJ1=JJ-1;YFRAC(JJ)=YFRAC(JJ1)+DYFR;DY=DY*AA;DYFR=DY/YVLAST
ENDDO
YFRAC(NY-1)=0.95
YFRAC(NY)=1.0

GROUP 7. Variables stored, solved & named
SOLVE(P1,U1,V1)

GROUP 9. Properties of the medium (or media)

** setting flow properties - density, viscosity, velocity

REL=100.
GROUP 13. Boundary conditions and special sources

** boundary conditions on the domain:
- ambient pressure on the outer boundary
- inflow at specified velocity from the left half
- a solid boundary surrounding the inner cylinder
- outflow at a solved velocity on the right half
- application of D'Arcy's Law in the inner cylinder, with "DARCI = v/k"

PATCH(PINF,NORTH,1,NX,NY,NY,1,1,1,1)
COVAL(PINF,P1,1.0E+05,0.0)
PATCH(PINF,CELL,1,NX,NY,NY,1,NZ,1,1)
COVAL(PINF,P1,1.0E5,0.0)
COVAL(PINF,U1,ONLYMS,0.0);COVAL(PINF,V1,ONLYMS,0.0)

PATCH(UPOL,CELL,1,NXF,NY,NY,1,NZ,1,1)
COVAL(UPOL,U1,FIXVAL,GRND1)
PATCH(UPOL2,CELL,3*NXF+1,NX,NY,NY,1,NZ,1,1)
COVAL(UPOL2,U1,FIXVAL,GRND1)
PATCH(VPOL,CELL,1,NXF,NY-1,NY-1,1,NZ,1,1)
COVAL(VPOL,V1,FIXVAL,GRND1)
PATCH(VPOL2,CELL,3*NXF+1,NX,NY-1,NY-1,1,NZ,1,1)
COVAL(VPOL2,V1,FIXVAL,GRND1)

PATCH(OUTFLOW,CELL,NXF+1,3*NXF,NY,NY,1,NZ,1,1)
COVAL(OUTFLOW,U1,ONLYMS,SAME);COVAL(OUTFLOW,V1,ONLYMS,SAME)

DARCI=1000
PATCH(DARCY,PHASEM,1,NX,1,NYP,1,NZ,1,1)
COVAL(DARCY,U1,DARCI,0.0)
COVAL(DARCY,V1,DARCI,0.0)

GROUP 15. Termination of sweeps
LSWEEP = 5000

GROUP 16. Termination of iterations
LITER(P1) = 12;LITER(U1)=1;LITER(V1)=1

GROUP 17. Under-relaxation devices
RELAX(U1,FALSDT,1.0);RELAX(V1,FALSDT,1.0)

GROUP 19. Special calls from EARTH to GROUND
POLRA=VINF

GROUP 20. Preliminary print-out
ECHO=T

GROUP 22. Spot-value print-out
NPLT=2;IXMON=NXF;IYMON=NYP+1
GROUP 23. Field print-out and plot control
NXPRIN=1;NYPRIN=1
PATCH(MAP,CONTUR,1,NX,1,NY,1,1,1,1);PLOT(MAP,P1,1.0,10.0)
PLOT(MAP,U1,1.0,10.0);PLOT(MAP,V1,1.0,10.0)

TSTSWP=1

STOP
*** PHOENICS input file for flow past a permeable cylinder near a solid wall

TALK=F;RUN(1,1);VDU=X11-TERM

** set to "body-fitted coordinates", in which a rectangular grid is fit to the specified geometry

BFC = T

** declaration of variables, including names of points in the domain

REAL(XC1,XC2,XC3,XC4,XC5,XC6,XC7,XC8)
REAL(YC1,YC2,YC3,YC4,YC5,YC6,YC7,YC8)
REAL(XP1,XP2,XP3,XP4,XP5,XP6,XP7,XP8,XP9,XP10)
REAL(XP11,XP12,XP13,XP14,XP15,XP16,XP17,XP18,XP19,XP20)
REAL(YP1,YP2,YP3,YP4,YP5,YP6,YP7,YP8,YP9,YP10)
REAL(YP11,YP12,YP13,YP14,YP15,YP16,YP17,YP18,YP19,YP20)

REAL(P1,TUPI,SQ2,ZRO)
REAL(DCYL,RCYL,DARCI)
REAL(NGAP,GAP,NBOXL,BOXL,NBOXT,BOXT,NBOXR,BOXR)
REAL(SPNX,SPANY,SPANZ)
REAL(NDL,NDR,NDT,NDG,NDM,NCX1,NCX2,NCY1,NCY2)

** setting constants, including NGAP = dimensionless cylinder-wall spacing

PI=2*ASIN(1.0);TUPI=2*PI;SQ2=SQR(2.0);ZRO=0.0
DCYL=0.01;RCYL=DCYL/2
NGAP=0.1
GAP=NGAP*DCYL
NBOXL=3
BOXL=NBOXL*DCYL
NBOXT=5
BOXT=NBOXT*DCYL
NBOXR=10
BOXR=NBOXR*DCYL
SPANX=BOXL+BOXR
SPANY=BOXT+GAP
SPANZ=DCYL

NDL=5*NBOXL
NDR=5*NBOXR
NDT=5*NBOXT
NDM=10
NDG=10

NX=NDL+NDG+NDR
NY=NDG+NDM+NDT
NZ=1

NCX1=1+NDL
NCX2=NDL+NDG
NCY1=1+NDG
NCY2=2*NDG
** definition of coordinates for points in the domain, where "XC" points delineate the cylinder

XCl=ZRO;YCl=ZRO
XCl=XCl-(RCYL/SQ2);YC2=YCl+RCYL*(1-1/SQ2)
XCl=XCl-RCYL;YC3=YCl+RCYL
XCl=XCl-RCYL;YC4=YCl+RCYL*(1+1/SQ2)
XCl=XCl;YC5=YCl+DCYL
XCl=XCl+(RCYL/SQ2);YC6=YC4
XCl=XCl+RCYL;YC7=YC3
XCl=XCl+(RCYL/SQ2);YC8=YC2

XP1=XCl-BOXL;YP1=YCl-GAP
XP2=XC2;YP2=YP1
XP3=XC8;YP3=YP2
XP4=XCl+BOXR;YP4=YP3
XP5=XP1;YP5=YC2
XP6=XP4;YP6=YP5
XP7=XP5;YP7=YC4
XP8=XP6;YP8=YC6
XP9=XP7;YP9=YCl+BOXT
XP10=XC4;YP10=YP9
XP11=XC6;YP11=YP9
XP12=XP8;YP12=YP9

** definition of points at the coordinates specified

GSET(P,C1,XC1,YC1,0.0)
GSET(P,C2,XC2,YC2,0.0)
GSET(P,C3,XC3,YC3,0.0)
GSET(P,C4,XC4,YC4,0.0)
GSET(P,C5,XC5,YC5,0.0)
GSET(P,C6,XC6,YC6,0.0)
GSET(P,C7,XC7,YC7,0.0)
GSET(P,C8,XC8,YC8,0.0)

GSET(P,P1,XP1,YP1,0.0)
GSET(P,P2,XP2,YP2,0.0)
GSET(P,P3,XP3,YP3,0.0)
GSET(P,P4,XP4,YP4,0.0)
GSET(P,P5,XP5,YP5,0.0)
GSET(P,P6,XP6,YP6,0.0)
GSET(P,P7,XP7,YP7,0.0)
GSET(P,P8,XP8,YP8,0.0)
GSET(P,P9,XP9,YP9,0.0)
GSET(P,P10,XP10,YP10,0.0)
GSET(P,P11,XP11,YP11,0.0)
GSET(P,P12,XP12,YP12,0.0)

** definitions of lines/arcs to connect points, as well as divisions on lines to specify the grid

GSET(L,CC1,C8,C2,NDG,1.0,ARC,XC1,YC1,0.0)
GSET(L,CC2,C2,C4,NDG,1.0,ARC,XC3,YC3,0.0)
GSET(L,CC3,C4,C6,NDG,1.0,ARC,XC5,YC5,0.0)
GSET(L,CC4,C6,C8,NDG,1.0,ARC,XC7,YC7,0.0)
GSET(L,LB1,P2,P1,NDL,1,0,0,0)
GSET(L,LB2,P3,P2,NDG,1,0,0,0)
GSET(L,LB3,P4,P3,NDR,1,0,0,0)
GSET(L,LB4,C2,P5,NDL,1,0,0,0)
GSET(L,LB5,C4,P7,NDL,1,0,0,0)
GSET(L,LB6,P9,P10,NDL,1,0,0,0)
GSET(L,LB7,P10,P11,NDG,1,0,0,0)
GSET(L,LB8,P11,P12,NDR,1,0,0,0)
GSET(L,LB9,C6,P8,NDR,1,0,0,0)
GSET(L,LB10,C8,P6,NDR,1,0,0,0)
GSET(L,LB11,P1,P5,NDG,1,0,0,0)
GSET(L,LB12,P5,P7,NDG,1,0,0,0)
GSET(L,LB13,P7,P9,NDT,1,0,0,0)
GSET(L,LB14,P12,P8,NDT,1,0,0,0)
GSET(L,LB15,P8,P6,NDG,1,0,0,0)
GSET(L,LB16,P6,P4,NDG,1,0,0,0)
GSET(L,LB17,P2,C2,NDG,1,0,0,0)
GSET(L,LB18,P3,C8,NDG,1,0,0,0)
GSET(L,LB19,C4,P10,NDT,1,0,0,0)
GSET(L,LB20,C6,P11,NDT,1,0,0,0)

** joining the lines in "frames" to be meshed

GSET(F,F1,P1,-,P5,-,C2,-,P2,-)
GSET(F,F2,P2,-,C2,-,C8,-,P3,-)
GSET(F,F3,P3,-,C8,-,P6,-,P4,-)
GSET(F,F4,P5,-,P7,-,C4,-,C2,-)
GSET(F,F5,C8,-,C6,-,P8,-,P6,-)
GSET(F,F6,P7,-,P9,-,P10,-,C4,-)
GSET(F,F7,C4,-,P10,-,P11,-,C6,-)
GSET(F,F8,C6,-,P11,-,P12,-,P8,-)
GSET(F,F9,C2,-,C4,-,C6,-,C8,-)

** setting the domain size

GSET(D,NX,NY,NZ,SPANX,SPANY,SPANZ)

** matching or fitting the meshed frames to the rectangular grid

GSET(M,F1,+J+I,1,1,1,LAP3)
GSET(M,F2,+J+I,1+NDL,1,1,LAP3)
GSET(M,F3,+J+I,1+NDL+NDG,1,1,LAP3)
GSET(M,F4,+J+I,1+NDL+NDG,1,LAP3)
GSET(M,F5,+J+I,1+NDL+NDG,1+NDG,1,LAP3)
GSET(M,F6,+J+I,1+2*NDG,1,LAP3)
GSET(M,F7,+J+I,1+NDL+NDG,1+2*NDG,1,LAP3)
GSET(M,F8,+J+I,1+NDL+NDG,1+2*NDG,1,LAP3)
GSET(M,F9,+J+I,1+NDL,1+NDG,1,LAP3)

** copying the planar mesh and displacing it some amount for unit depth

GSET(C,K2,F,K1,1,NX,1,NY,+,0,0,SPANZ)
GROUP 1. Run title and other preliminaries
TEXT(WALL/POURE CYLINDER, RE = 1000
TITLE
REAL(VCLSTR)
REAL(REL)

GROUP 7. Variables stored, solved & named
SOLVE(P1,U1,V1)

GROUP 9. Properties of the medium (or media)

** setting flow properties - density, viscosity, velocity

REL=1000.
RH01=1.20;ENUL=1.5E-5
VCLSTR=REL*ENUL/DCYL

GROUP 13. Boundary conditions and special sources

** boundary conditions on the domain:
- wall velocity equal to flow inlet (for motion relative to body)
- inflow at specified velocity from the left side
- ambient pressure on the outer boundary
- outflow at a solved velocity on the top and right sides
- application of D'Arcy's Law in the inner cylinder, with "DARCI" = v/ka

PATCH(DOMWALL,SWALL,1,NX,1,1,1,NZ,1,1)
COVAL(DOMWALL,U1,1.0,VCLSTR)
COVAL(DOMWALL,V1,1.0,0.0)
PATCH(DOMWALL,CELL,1,NX,1,1,1,NZ,1,1)
COVAL(DOMWALL,U1,1.0,VCLSTR)
COVAL(DOMWALL,V1,1.0,0.0)

PATCH(UAIRW.WEST,1,1,1,NY,1,NZ,1,1)
COVAL(UAIRW,U1,1.0,VCLSTR)
COVAL(UAIRW,V1,1.0,0.0)
PATCH(UAIRW,CELL,1,1,1,NY,1,NZ,1,1)
COVAL(UAIRW,U1,1.0,VCLSTR)
COVAL(UAIRW,V1,1.0,0.0)

PATCH(PINFW.WEST,1,1,1,NY,1,NZ,1,1)
COVAL(PINFW,P1,1.0E05,0.0)
PATCH(PINFW,CELL,1,1,1,NY,1,NZ,1,1)
COVAL(PINFW,P1,1.0E05,0.0)

PATCH(PINFN,NORTH,1,NX,NY,1,NZ,1,1)
COVAL(PINFN,P1,1.0E05,0.0)
PATCH(PINFN,CELL,1,NX,NY,1,NZ,1,1)
COVAL(PINFN,P1,1.0E05,0.0)

PATCH(PINFE,EAST,NX,NX,1,NY,1,NZ,1,1)
COVAL(PINFE,P1,1.0E05,0.0)
PATCH(PINFE,CELL,NX,NX,1,NY,1,NZ,1,1)
COVAL(PINFE,P1,1.0E05,0.0)
PATCH(OUTFLOW,CELL,NX,NX,1,MY,1,NZ,1,1)
COVAL(OUTFLOW,U1,ONLYMS,SAME)
COVAL(OUTFLOW,V1,ONLYMS,SAME)

PATCH(OUTFLOW,CELL,1,NX,NY,NY,1,NZ,1,1)
COVAL(OUTFLOW,U1,ONLYMS,SAME)
COVAL(OUTFLOW,V1,ONLYMS,SAME)

DARCI=1000.
PATCH(DARCY,PHASEM,NCX1,NCX2,NCY1,NCY2,1,NZ,1,1)
COVAL(DARCY,U1,DARCI,0.0)
COVAL(DARCY,V1,DARCI,0.0)

GROUP 15. Termination of sweeps
LSWEEP = 2000

GROUP 16. Termination of iterations
REAL(MAXV,MINV,FDT); MAXV=VCLSTR; MINV=0.18/100.
FDT=0.15*MINV/MAXV
LITER(P1)=12;LITER(U1)=1;LITER(V1)=1

GROUP 17. Under-relaxation devices
RELAX(U1,FALSDT,FDT);RELAX(V1,FALSDT,FDT)

GROUP 20. Preliminary print-out
ECHO=T

GROUP 22. Spot-value print-out
NPLT=2;DMON=NDL+NDG/2;IMON=NDG/2

GROUP 23. Field print-out and plot control
NXPRIN=1;NYPRIN=1
PATCH(MAP,CONTRB,1,NX,1,MY,1,1,1,1);PLOT(MAP,P1,1.0,10.0)
PLOT(MAP,U1,1.0,10.0);PLOT(MAP,V1,1.0,10.0)
ITABL=3

TSTSWP=-1

STOP