

CFD Modelling in the Cement Industry

Victor J. Turnell, P.E.,
Turnell Corp., USA,
describes computational
fluid dynamics (CFD)
simulation and its
benefits in applications
in the cement industry.

Introduction to CFD

Engineers are increasing the use of CFD simulation in the design process to validate and optimise designs. The growth in CFD usage is a result of the increasing processing power of computers and improved CFD simulation software. CFD is a branch of fluid dynamics that uses numerical methods to analyse fluid flows. It estimates and predicts flow velocity, temperature, pressure, density, chemical concentrations and many other properties for any region in the flow profile. It is a virtual modelling technique with powerful visualisation capabilities.

One of the benefits of using CFD simulation is that it is significantly less costly to test various operating scenarios and designs than completing physical tests onsite or with scale models. It also gives the user additional insight into the operations beyond what is available with process instrumentation, such as thermocouples and pressure sensors. These types of sensors provide data only for a point in the fluid flow. Outputs from the CFD simulations include contour diagrams showing various data, such as temperatures and chemical concentrations throughout any plane in the fluid. Path lines can be generated that show fluid flow data such as velocity (speed and direction) throughout the fluid flow. In addition, videos can be generated to show the transient and cyclical flow behaviours.



Table 1. Operating scenarios

	Mass flow rate (kg/s)		Gas temperatures
	Operation scenario 1	Operation scenario 2	Both scenarios (°C)
Preheater 1 flow	49	49	269
Preheater 2 flow	37	37	200
Total preheater gases	86	86	-
Raw mill bypass flow	23	7	Varies
Ambient air flow	32	16	14
Difference	9	9	-
Mill inlet 1	47.5	47.5	Varies
Mill inlet 2	47.5	47.5	Varies
Total	95	95	-

Applying CFD

The general steps in applying CFD simulation are as follows:

1. Define the geometry of the fluid.
2. Divide the volume occupied by the fluid into discrete cells (meshing).
3. Define the physical models, such as the equations of continuity, energy balances and turbulence models applicable to the simulation.
4. Define the cells and boundary conditions.
5. Run the simulation that solves the equations iteratively.
6. Analyse the results of the simulation.

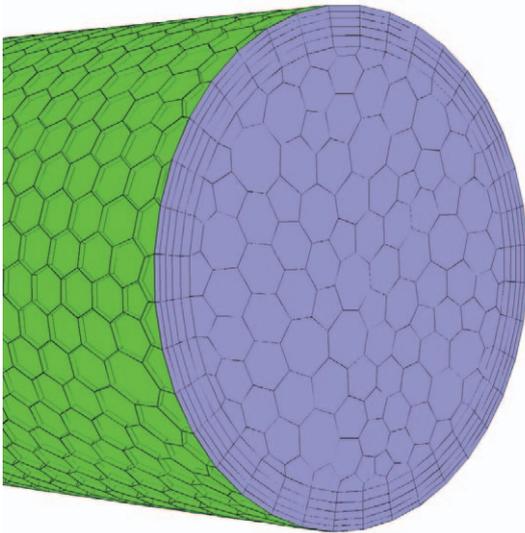


Figure 1. Polyhedron meshing example.

Usually, the CFD software package includes software that can be used to model the fluid geometry. The geometry of the fluid can also be generated with computer aided design (CAD) software such as Autodesk AutoCAD® or Inventor® and Dassault Systèmes Solidworks®. Furthermore, the CAD software can also be used to produce the fabrication drawings of the system; for example, the fabrication drawings of a duct. The advantage of using this type of software is that the geometry is generated once; thereby avoiding some potential errors caused by creating the geometry more than once. For example, a potential error could be to run the CFD simulation using a fluid geometry that does not correspond to the geometry used in the design and installation.

There are several types of meshing and cells available. They include triangles and quadrilaterals for two-dimensional applications. For three-dimensional applications, there are tetrahedrons, hexahedrons, prisms/wedges, pyramids and polyhedrons. Figure 1 shows a polyhedron mesh. Other mesh types are available that have speciality applications. The mesh selection depends on the application. The selection should consider the setup time, computational processing time required to obtain a solution and the numerical diffusion. The quantity and quality of cells generated affects these values. Numerical diffusion is one of the dominant sources of error in multidimensional situations. Good CFD simulating practice establishes the independence of the solution relative to the type and number of cells used in the model.

Numerous CFD models are available. They can take into account continuity, energy balances, multi-phases including gases, liquids and solids, various types of turbulence models, radiation, heat transfer, different mixing and transport of chemical species, chemical reactions and combustion, pollutant formation, discrete phases (particles), and sliding and dynamic meshes. The models have different levels of complexity that may be applicable to some cases and not to others. The more models included in the simulation, the more computationally time consuming the solutions will become. The engineer can also make simplifying assumptions that reduce the computational processing

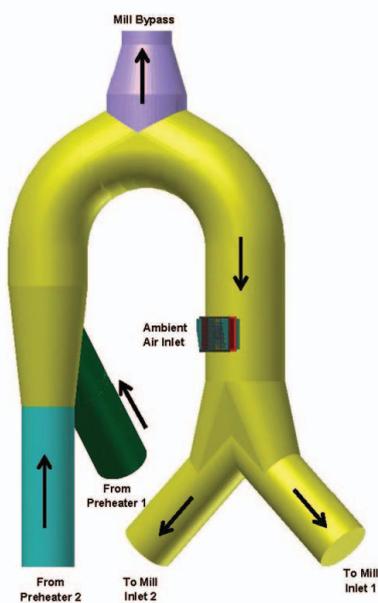


Figure 2. Existing duct configuration.

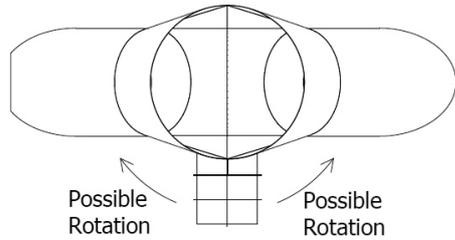


Figure 3. Potential ambient air inlet position.

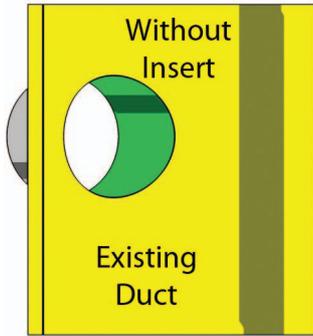


Figure 4. Potential ambient air inlet position.

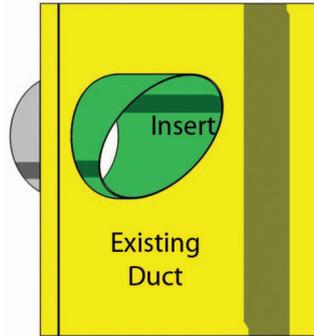


Figure 5. Insert design considered.

Case study 1 – gas temperature distribution

Introduction

This case study considers a retrofit of a duct system at a cement plant. The objective of the retrofit is to blend ambient air with hot gases going to a vertical roller mill for temperature control. The objective of the CFD analysis was to minimise the temperature difference between the two streams of gases entering the vertical roller mill. The geometry and fabrication drawings were generated in Autodesk Inventor®. The CFD simulations were completed with Ansys Fluent®.

The starting point of this analysis is the two streams of hot gases originating from the two preheater towers. Initially, these gas streams are combined into one. Later, these streams are split into two: one going to the mill and the other bypassing the mill. The gas stream going to the mill is mixed with ambient air for temperature control of the gases to the mill. Figure 2 shows the existing duct configuration. The red and cyan duct section is the new ambient air inlet duct; the green duct section is from preheater tower 1; the cyan duct section is from preheater tower 2 and the purple duct section is the mill bypass duct.

Two operating scenarios were analysed to ensure that the gas temperature difference was maintained within acceptable limits for the range of operating conditions. Operating scenario 1 considers the maximum ambient air flow for cooling. Operating scenario 2 considers half the maximum ambient air flow for cooling. Table 1 summarises the flows.

One variable considered in the analysis is the location of the ambient air duct radially around the main duct, as shown in Figure 3. The elevation was fixed as shown in Figure 2. The other variable was whether or not to use an insert to induce mixing. Figure 4 shows the ambient air inlet duct without an insert whilst Figure 5 shows the duct with the insert. Other options exist, such as the introduction of the ambient air earlier in the duct to allow some time for mixing before splitting the gases. However, these options were eliminated because of layout and capital cost constraints.

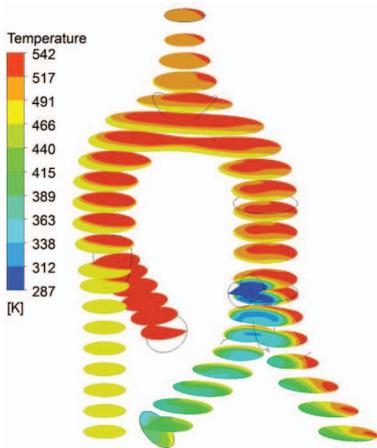


Figure 6. Temperature contours for the initial design, scenario 1.

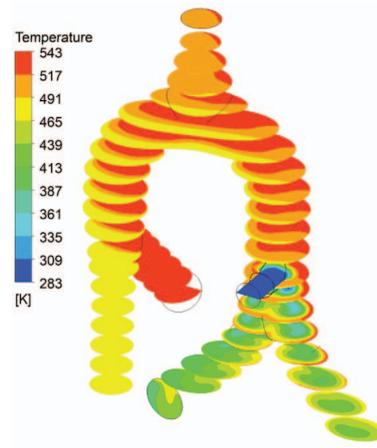


Figure 7. Temperature contours for the selected design, scenario 1.

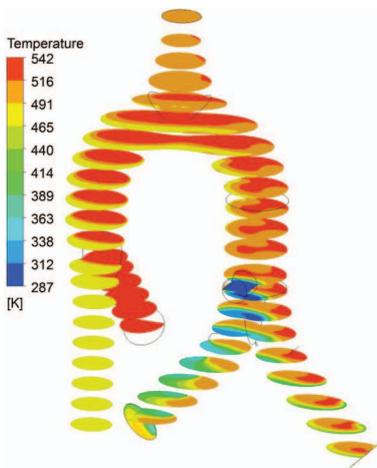


Figure 8. Temperature contours for the initial design, scenario 2.

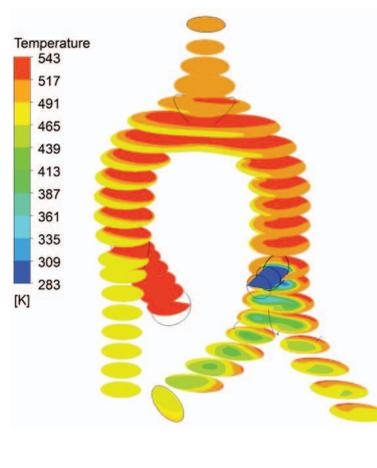


Figure 9. Temperature contours for the selected design, scenario 2.

Initial design

The initial design was a duct located without the insert. For operating scenarios 1 and 2, the differences in average gas temperature of the two gas streams entering the vertical roller mill were 52 °C and 45 °C, respectively. These temperature differences were considered unacceptable. Figures 6 and 8 show temperature contours of the gases for operating scenarios 1 and 2, respectively. Another noteworthy observation is the temperature profile of the gases in the duct. The gas streams from the two preheater towers were not mixing rapidly. The yellow, red and orange colours on the temperature contours confirm the stratification of the two gases.

Selected design

The selected design was a duct located with the insert. For operating scenarios 1 and 2, the temperature differences between the two gas streams entering the raw mill were 21 °C and 18 °C, respectively, which were considered acceptable. Figures 7 and 9 show temperature contours of the gases for operating scenarios 1 and 2, respectively.

Summary

In this case study, the determination of the location for the ambient air inlet duct would probably have been done by experience and trial and error. The decision to install the duct insert would probably not have been considered until after operating the system and experiencing a significant temperature difference. The results of this approach would most likely have caused mechanical damage to the mill components or caused an extended plant shutdown, resulting in increased costs.

CFD simulations provided the information and insight to the duct designer to optimise the system design before implementing the modifications. In this case, CFD simulation allowed the designer to reduce the variance from 52 °C to 21 °C in operating scenario 1 and from 45 °C to 18 °C in operating scenario 2. The temperature distribution inside the duct during the various operating conditions would not have been known unless actual measurements were taken onsite or by completing the CFD simulation. In addition, CFD simulation was used to predict the temperature difference once the retrofit was completed. This significant advantage would not otherwise have been possible.

Case study 2 – gas velocity distribution and pressure drop

Introduction

This case study considers a modification to a duct system at a cement plant. The existing plant has two preheater towers that are vented through two fabric filters. Each preheater tower has its own duct to its designated fabric filter. The objective of this modification was to eliminate one of these ducts and install a crossover duct to maintain use of both existing fabric filters. A portion of the remaining existing duct would be used for gases from both preheater towers. A second phase to this project will be to replace the portion of the duct used for gases from both preheater towers with one that has a larger diameter.

Figure 10 shows the general arrangement of the initial duct design. Once again, the geometry and fabrication drawings were generated in Autodesk Inventor® and the CFD simulations were completed with Ansys Fluent®.

Initial design

The maximum gas velocity in the takeoff for the initial design was estimated to be 22 m/s, based on a plug flow approach. This approach considers an even velocity distribution across the cross-sectional area of the takeoff. The pressure drop

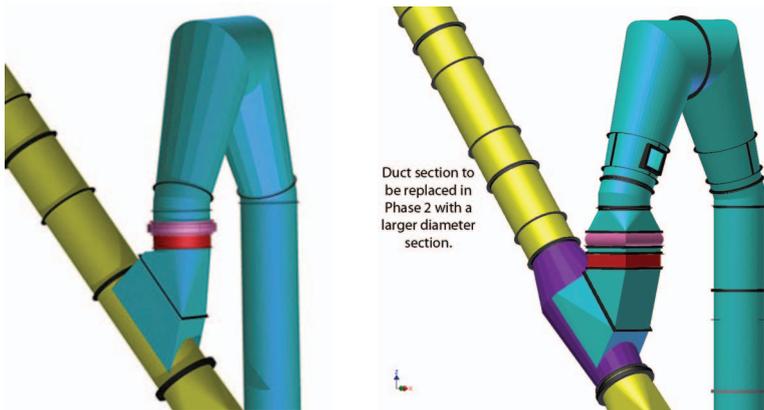


Figure 10. Original duct arrangement.

Figure 11. Final duct arrangement.

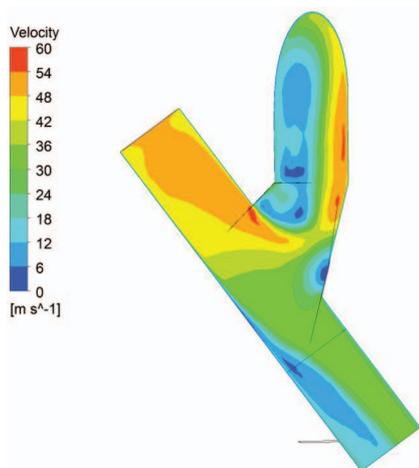


Figure 12. Gas velocities in the initial duct takeoff design.

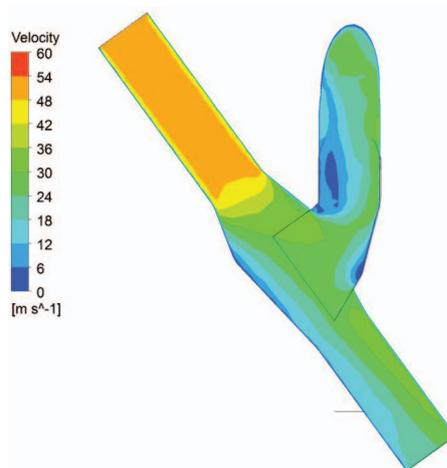


Figure 13. Gas velocities in the selected duct takeoff design.

across the takeoff would have been estimated using a conventional approach, such as estimating the pressure drop across the takeoff by multiplying the velocity pressure with the local loss coefficient available for 'standard' takeoff arrangements. Since the initial takeoff design is not conventional, the designer's judgment would have been used to estimate the loss coefficient factor while considering the data available for the 'standard' takeoff arrangements. The estimated pressure drop would not have been reflective of the pressure drop experienced during actual operation based on the CFD simulation performed. The initial takeoff design would have been acceptable based on the estimated velocity and pressure drop initially calculated without the CFD simulation.

The CFD simulation of the initial design indicated a maximum gas velocity of 54 m/s in the takeoff. It also indicated a pressure drop of 760 Pa between the existing duct inlet and the new duct exit. Figure 12 shows a velocity contour on a plane passing through the takeoff. Since the gas stream contains entrained particulates, this velocity would most likely have resulted in a relatively high wear rate of the duct, so this design was considered to be unacceptable.

Selected design

A portion of the existing duct was replaced with a larger diameter transition section to reduce the gas velocities and pressure drop, and decrease the wear rate. Figure 11 shows the general arrangement of the selected duct design. The yellow duct is the existing duct, the cyan duct is the new duct, whilst the purple duct and the redesigned takeoff indicate the modification between the initial design and the selected design.

A CFD simulation of this new design indicated a maximum gas velocity of 28 m/s in the takeoff. It also indicated a pressure drop of 330 Pa between the existing duct inlet and the new duct exit. Figure 13 shows a velocity contour on a plane passing through the takeoff. Although the maximum gas velocity is relatively high, this design was considered acceptable for the first phase of this project.

Summary

In this instance, it is likely that the plant personnel would have operated the system within the operating conditions, as indicated by the CFD simulation of the initial design. This design would have caused

higher velocities, higher pressure-drop, greater power consumption and an increased wear rate of the takeoff as compared to the selected design. At a future date, the personnel would probably have had to redesign and replace the takeoff. Operating the system with the initial design would have increased the operating costs, as well as the maintenance cost of the system. If the plant were to redesign and replace the takeoff, it would have resulted in additional cost expenditure.

CFD simulations provided the information for the duct designer to optimise the system design before implementing the modifications. The maximum gas velocity decreased from 54 m/s to 28 m/s and the pressure drop decreased from 760 Pa to 330 Pa by applying the design modifications. This decrease in velocity significantly lowers the wear rate of the duct and the pressure drop reduction results in an estimated 28 kW power consumption savings, depending on the fan flow control method.

Conclusion

In both case studies, Turnell Corp. completed all phases of process design, CFD simulation and engineering. There is a significant advantage in using one company for all of these procedures that understands the implications of different design configurations on the cement manufacturing operations, operating costs and capital costs. This is especially important since the selected design is usually a compromise among operational requirements, capital costs and operating costs. This was instrumental in the system optimisation process. Turnell Corp.'s cement process experience also served as a reality check to the CFD simulations' results. Cement plant personnel are encouraged to pursue companies that combine CFD simulation, cement process and engineering expertise to obtain optimal solutions. The cement industry has many applications where CFD simulation can assist in optimising the plant design and operation. These include:

- Calciners.
- Cyclones in preheater towers.
- Burners.
- Kiln/clinker cooler.
- Gas conditioning.
- Fabric filter and electrostatic precipitators.
- Ducts. 🌐