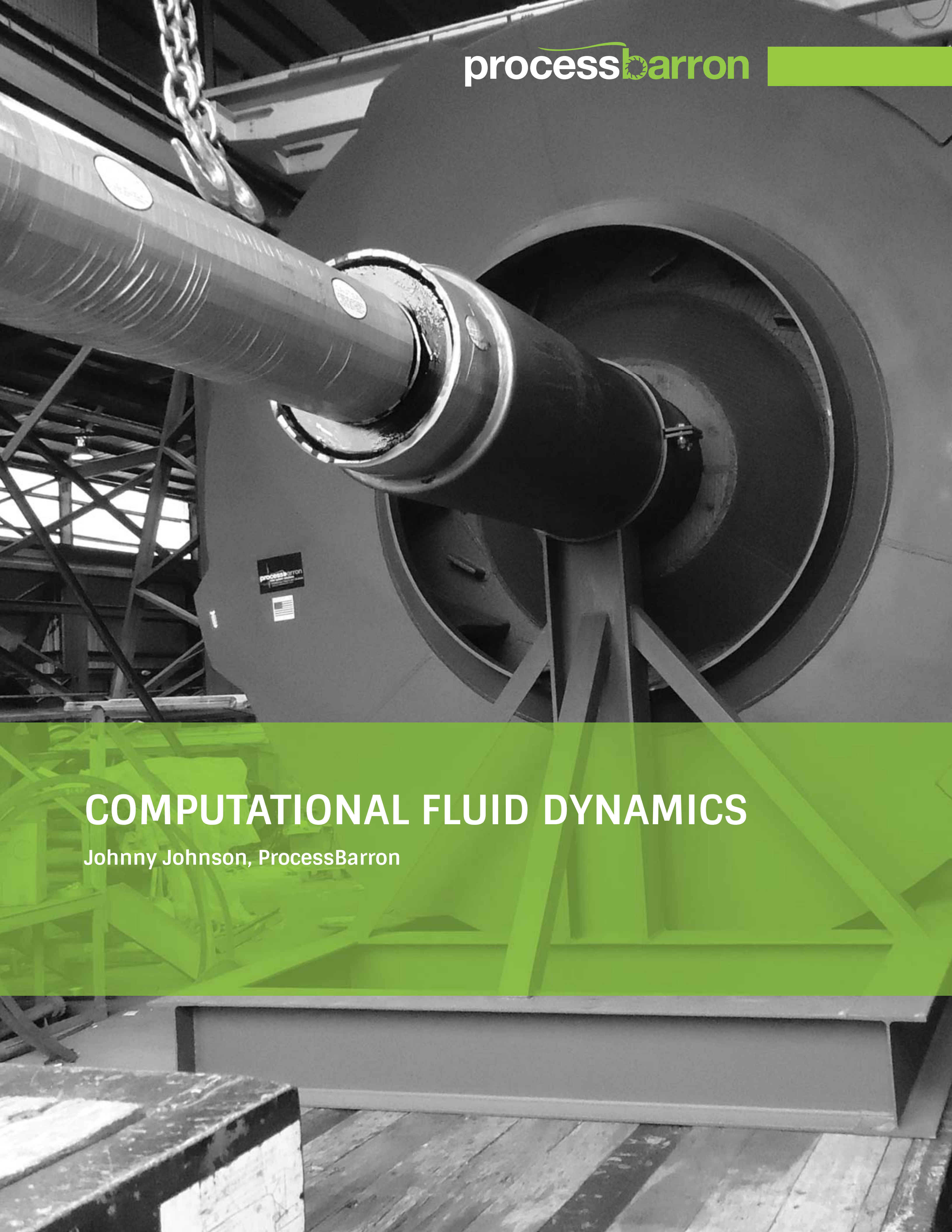

COMPUTATIONAL FLUID DYNAMICS

Johnny Johnson, ProcessBarron

SYNOPSIS

Computational Fluid Dynamics (CFD) are an effective way to assess the viability of a fan installation. A form of computer simulation, CFD is much less expensive than scale model testing and provides an opportunity to test design variations iteratively in a virtual environment so that one can arrive at a fully optimized design. This case study shows how CFD proved invaluable for evaluating an excessive buildup problem in a cement plant air handling system and creating a fan design that resolved the issue and increased the overall efficiency of the impeller.

Computational fluid dynamics

**by Johnny Johnson,
Process Barron, USA**

Finite Element Analysis (FEA) software has been commercially available for over 30 years. Some of the first areas to be studied using this method were structural and airframe design.

In essence, FEA provides a mathematically stable method for numerically solving systems of partial differential equations that cannot be evaluated by analytical methods. Although the numerical solution offers only an approximation of the correct answer, it can be driven very close to the correct solution in many cases.

The partial differential equations that describe the behaviour of simple structural elements with straight-forward boundary conditions, such as simply-supported beams and plates, have analytical solutions for many loading conditions. The general equations that describe fluid flow – the Navier-Stokes equations – have no known analytical solution and are much more difficult to solve numerically than the equations defining the behaviour of simple structures.

Due to recent advances in computing power and three-dimensional graphical modelling, computational fluid dynamics (CFD) has become a practical tool for industrial applications. Design engineers using commercially available CFD software and desktop computers can now study flow problems that once required the devoted attention of university researchers and the power of supercomputers. The subject of this article is the use of CFD to redesign a cement kiln ID fan impeller that experienced a common problem in cement plant fan applications: excessive buildup of material on the fan blades leading to high levels of vibration.

The initial desire of the cement plant owner was to replace the inefficient 1980s model radial-bladed fan design with a higher-efficiency backward curved blade design to reduce the power consumption of the fan. The potential power savings of over 200hp looked very attractive. The fan vendor initially proposed to provide a complete new fan assembly – both housing and impeller – to meet the design requirements, but discovered that the existing fan foundation would have to be substantially altered to accommodate the new housing. In cooperation with the plant owner, the fan vendor undertook a series of scale-model studies to measure the flow and pressure performance of the higher efficiency backwardly curved impeller design operating in the existing radial blade fan housing.

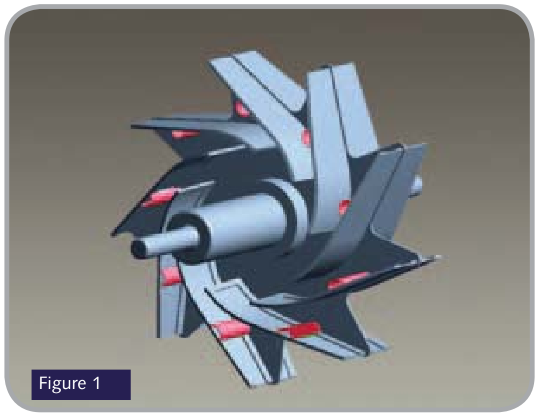

Figure 1

Prior to the model studies, the fan vendor used FEA to perform a structural analysis of the impeller to design it for the 420mph peripheral velocity and 725°F operating conditions. One of the structural design features that was needed to reduce the stresses at the maximum speed was a half-pipe stiffener attached to the back of the blade (see Figure 1). The scale model flow studies incorporated this stiffening feature and still achieved the plant's minimum efficiency objectives. The full-scale impeller was constructed and placed in service. And, although the power savings were immediately obvious, the fan soon began to experience significant vibration problems.

The plant's maintenance staff quickly traced the vibration problems to large accumulations of kiln feed material in the region upstream of the half-pipe stiffener on the back of the fan blade (see Figure 2). The plant's observations of this buildup made it clear that the material

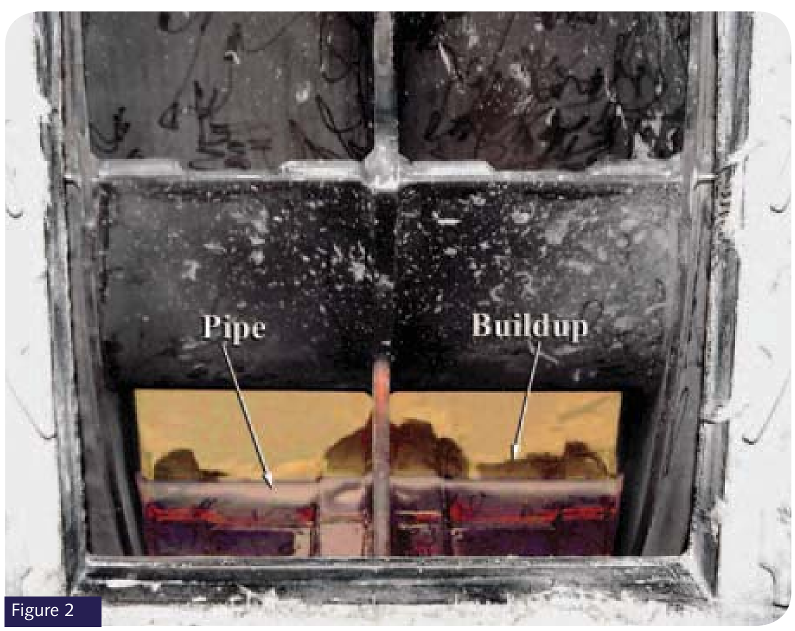

Figure 2

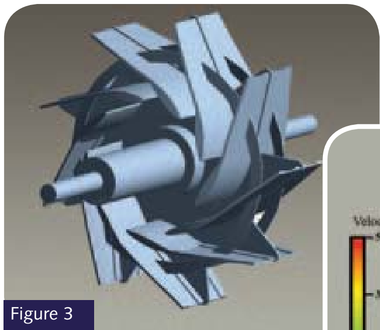
Figure 3


Figure 5

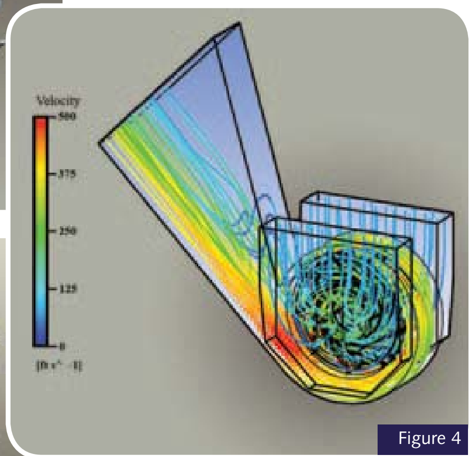

Figure 4

was not adhering to the impeller, but was held in place by centrifugal force alone. As soon as the impeller came to rest, all of the buildup would simply fall off. Due to production demands, the plant was forced to remove the impeller from service. The cement plant management wanted to redesign the impeller in such a way that the original efficiency was not compromised, the structural integrity was preserved, and the buildup problem was eliminated.

The redesign team decided to take advantage of the fact that the kiln feed material handled by this kiln ID fan had not produced any noticeable wear on the impeller, even after many years of service. So, two new features were incorporated in the modified design an airfoil section at the leading edge of the blade, and a stiffening ring in the centre of the blade (see Figure 3). The airfoil section would replace the concave surface on the back of the blade with a convex surface that would be much less likely to trap buildup. The stiffening rings would replace the half-pipe stiffeners and also provide little opportunity for buildup to occur.

The finite element stress analysis demonstrated that these features created a structural design that was superior to the initial design. The elimination of the half-pipe stiffener and the addition of the airfoil nose section certainly appeared to address the buildup concerns, but to persuade the owner that the impeller would not experience another episode of buildup related trouble, a more exact understanding was needed before it was returned to service.

With the original scale model studies as their basis, the redesign engineering team used the ANSYS CFX software to model the flow and pressure performance (see Figure 4). After several model refinements, the performance characteristics predicted by the CFD model were very close to the measurements taken in the scale model studies (see Table 1). As part of the flow studies, a particle size analysis was performed on a sample of the kiln feed material that had accumulated on the back of the fan blades. The resulting particle size distribution was used to model the flow of suspended kiln feed material though the kiln ID fan. Figure #5 illustrates that the inner portion of the blade, upstream of the half pipe stiffener, is where most of the kiln feed material was impacting the blade, causing the troublesome buildup. The stiffener acted as a dam, blocking the flow of suspended particles along the back of the blade.

Table 1: scale model studies

Model	*Total pressure (in wg)*	*Fan power (bhp)*
Scale model	30.05	2045
CFD model	29.41	2063
% difference	-2.1	0.9

The CFD model showed good correlation to both the flow characteristics and the suspended kiln feed material behaviour. This gave the engineers confidence that a similar CFD model of the modified version of the impeller would provide accurate predictions of its behaviour.

The modified impeller geometry was subjected to the same flow and suspended material conditions as the initial design. The results were noticeably better (see Figure 6). The new geometry created a much better distribution of suspended material flow over the back of the blade. The particle impact locations shifted away from the inner portion of the blade surface and toward the perimeter, where the possibilities for accumulation are very limited.

As an added bonus, the airfoil section improved the overall efficiency of the impeller, promising an even greater reduction in power consumption than the initial design. These results persuaded the owner to return the impeller to service.

Another benefit of this approach was the considerable cost savings achieved by using CFD to perform the flow analyses instead of scale model testing – savings measured both in terms of expense and time. CFD allows many modifications to be tested by simply altering the geometry of an electronic model and re-running the solution processor. To accomplish the same changes on a physical test model commonly requires hours of fabrication and performance testing, and occasionally, a completely new model.

In summary, this simple case study demonstrates the practical use of CFD in a real-world cement plant application.

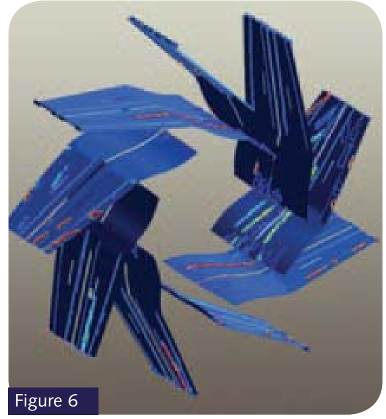
Figure 6