

# Computational Fluid Dynamic Modeling Aids In the Development of a Novel Spray Drying Technology

## Written by:

Timothy J. Fry, Ph.D.  
Stress Engineering Services, Inc.  
Process Technology Group  
5380 Courseview Drive  
Mason, Ohio 45040

(513) 336-6701 (phone)  
(513) 336-6817 (fax)  
tjfry@ses-oh.com  
www.processinnovation.com  
www.stress.com

## Biography:

Dr. Fry is a Staff Consultant with Stress Engineering Services, Inc. in Cincinnati, Ohio. In his position he uses analytical, numerical, and experimental techniques to solve a wide variety of industrial and process flow and heat transfer problems. Most recently he has been working on process drying, including the CFD of spray dryers and development of a climate-controlled wind tunnel to measure the drying rates of a variety of products. He has bachelors and masters degrees in Mechanical Engineering from the University of Illinois along with a Ph.D. in Aeronautical Engineering from the University of Dayton.

## Abstract:

Computational Fluid Dynamics (CFD) is used to analyze the performance of an industrial spray dryer in advance of making major structural changes to the dryer. It was anticipated that the proposed change would greatly improve the quality of the product. Despite the potential improvement, the risk of lost profit during the changeover (especially if the improvement did not materialize) had to be minimized before the project could proceed. We used CFD in this instance to help minimize the risk and avoid unnecessary downtime during testing. Specifically, the CFD models helped determine equipment configuration and process settings. In the end, the validated CFD models predicted the baseline and modified dryer performance very well. In addition, the CFD results helped give the process engineer confidence that the proposed modification would work before capital equipment was ordered and field-testing scheduled with the plant.

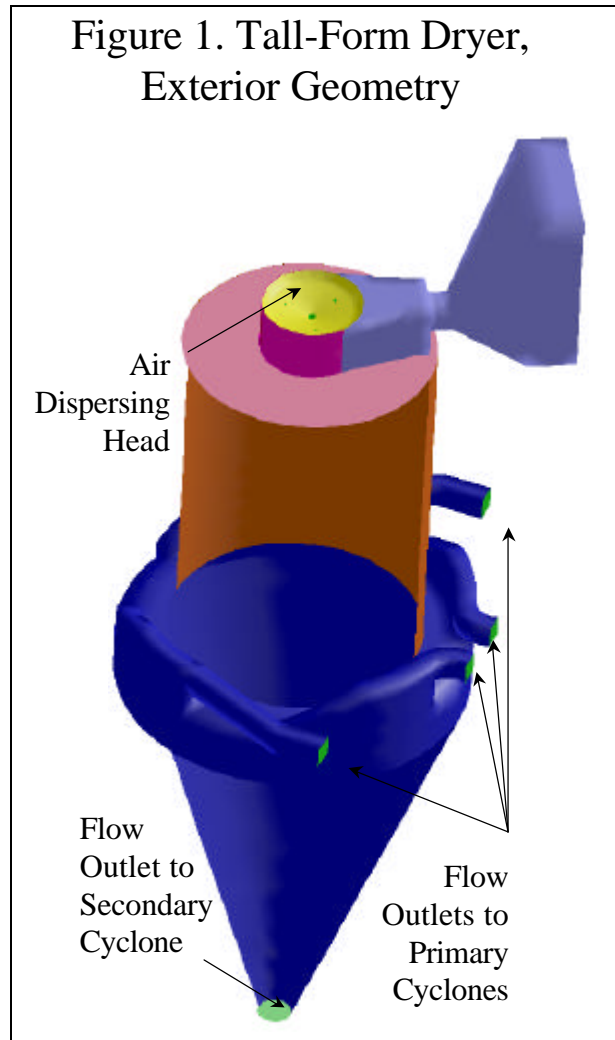
**This paper was published in two parts in the September (vol. 15, #9) and October (vol. 15, #10) 2001 issues of Powder and Bulk Engineering Magazine.**

**Problem Statement And Analysis Goals:**

In the powder spray drying industry, cost and efficiency considerations dictate that the manufacturing process be run at capacity and without interruption for as long as possible between cleaning cycles. The reasons for this fact are fairly straightforward. In the first place, the process is not profitable when it is shutdown. In addition, the transient system response during start-up uses raw materials and energy, but produces no salable product until the system reaches thermal and flow equilibrium.

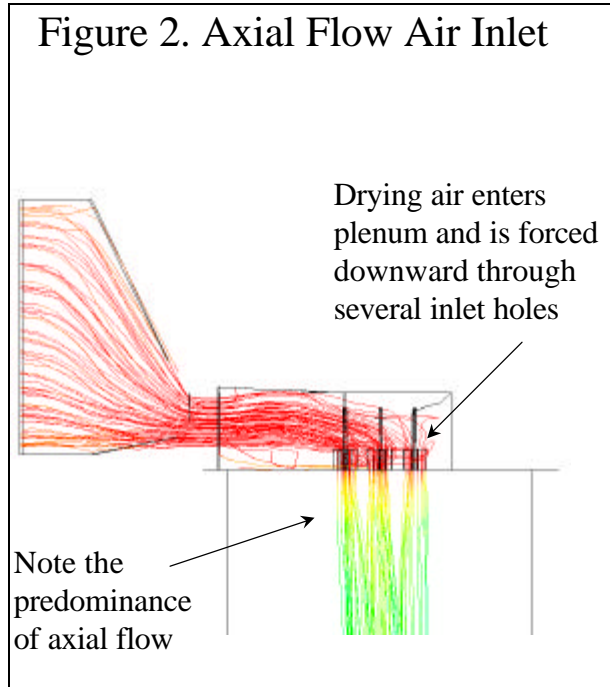
As with any high volume continuous manufacturing operation, the economics of the spray drying process are such that extended downtime, even for the purpose of installing efficiency and quality improvements, are avoided unless absolutely necessary. When these extended shutdowns do occur, they are rigorously planned and scheduled to minimize unexpected delays in bringing the system back into operation. In this environment, process modifications and improvements must work as designed, the first time. It is vitally important that the process engineers do proper front-end engineering and analysis of any processing or equipment change. Process modeling can help the process engineer design new equipment as

well as select preliminary operating parameters for the new process and equipment to assure a smoother and shorter return to normal production.



A valuable tool, used for the conceptual engineering analysis of spray dryer designs and process optimization in this project, is computational fluid dynamics (CFD) modeling. Using high-end UNIX workstations and commercially available CFD codes one can model many aspects of the drying process. In

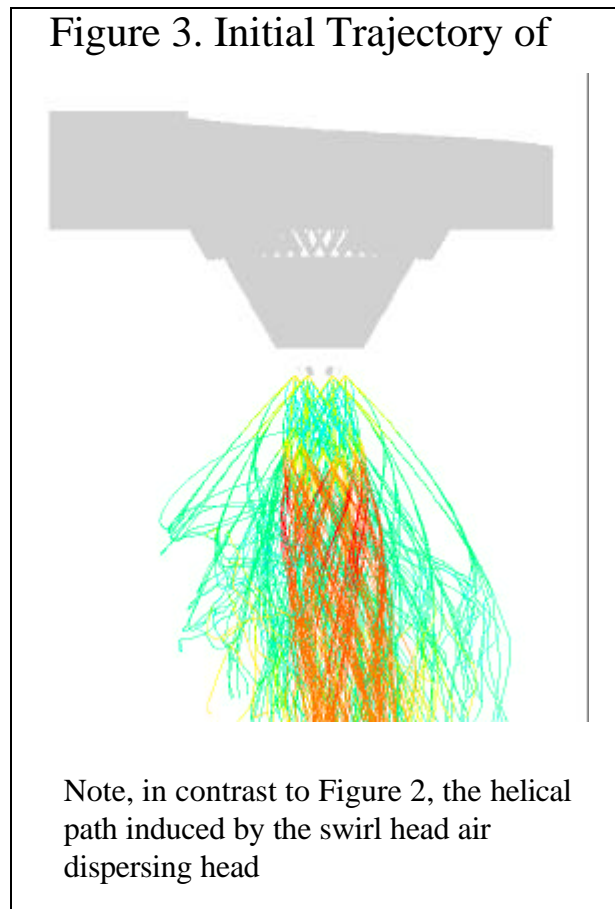
this example, CFD modeling was used to define a spray drying process for the manufacture of agglomerated milk based powders.



The details of the CFD models and cases studied will be discussed shortly. First, however, it must be emphasized that with a complicated system like a spray dryer, it is not suggested that the CFD model results were the sole basis of the final project go-no-go decision. Instead, the CFD modeling was pursued because there was strong evidence that such modeling could help the process engineer make more informed decisions and avoid costly mistakes and prolonged field trials. When combined with practical knowledge of the drying process, the model results were very beneficial in predicting the

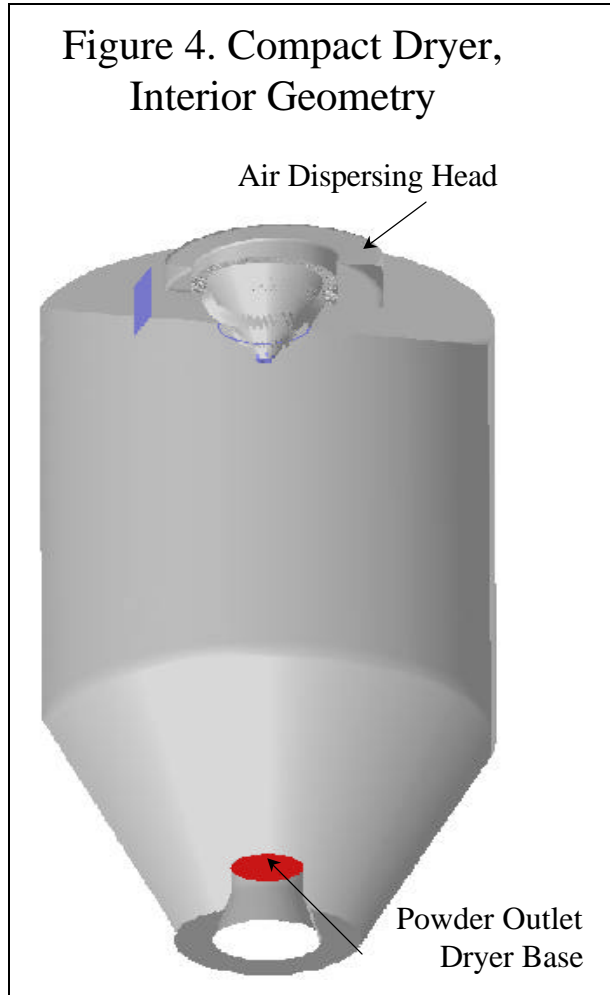
likely impact of the proposed design changes.

The specific project discussed in this article is the proposed conversion of a tall-form dryer (Figure 1.), whose predominant inlet flow pattern is axial (Figure 2.), to a revised air inlet configuration that generates a combination of axial and high swirl flow (Figure 3.).

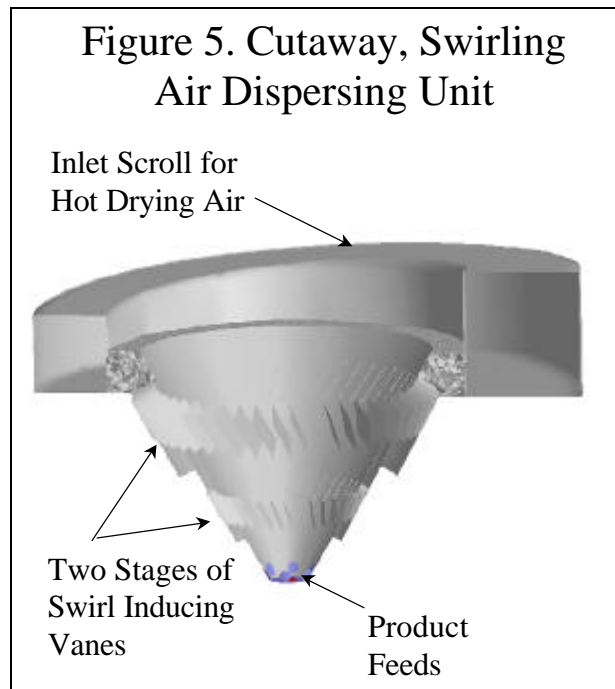


The baseline CFD model of the existing tall-form model was built to assure that the internal flow characteristics of the production dryer were accurately captured. In addition

to the CFD modeling of the tall-form dryer, CFD modeling of a wide-body dryer (Figure 4.) dryer with high swirl drying air was completed.



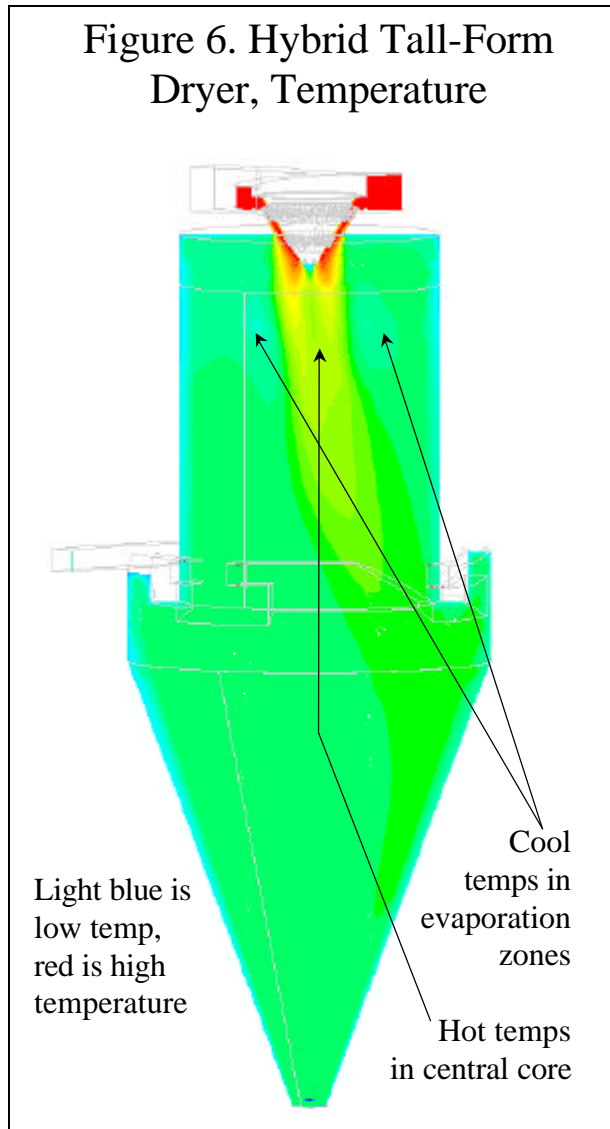
thermodynamic performance, product characteristics, and chamber fouling. Only after completing definitive field tests in the wide-body dryer and numerous CFD model runs with the proposed "hybrid" tall-form dryer design, was a production tall-form dryer modified and field-tested to determine its ability to produce a high quality agglomerated product and assess chamber fouling potential.



The modeling of the wide-body dryer was necessary to validate the CFD solution of the high-swirl air disperser (Figure 5.) and to provide the process engineers with a test bed for evaluating various "what if" scenarios without making any capital equipment changes to the dryers. Specifically, the engineers wanted to explore the effects of swirl velocity on dryer

Agglomeration and instantizing are used synonymously in the spray dried powder industry. The desired result of agglomeration is a free flowing, dust free and readily dissolvable powder. Examples of common agglomerated consumer products include instant breakfasts, powdered cocoa mixes, diet mixes, instantized milk powder and some infant formula powders. The

mechanism of agglomeration is deceptively simple in that particles are brought into contact with each other while they contain enough moisture to stick together and then the agglomerates are dried to the final moisture specification.



Instantizing can be accomplished in a stand-alone rewet agglomerator system or within a spray dryer. The rewet agglomeration process is ideal

for dry blended mixes, but the once-through approach is preferred for spray-dried powders because the capital and operating costs of the rewet system can be avoided. Unfortunately, the spray dryer process temperatures are significantly higher than those generated within the rewet agglomerator and can result in undesirable product quality defects. Figure 6 shows the temperatures within the dryer chamber. Note the hot core flow with cooler temperatures in the evaporation zones. It will be our goal to agglomerate the product in the cooler zones and create the proper combination of product moisture, temperature and particle concentrations during agglomeration.

Operating experience with both styles of dryer has demonstrated greater ability to adjust the critical parameters for agglomeration with the high swirl airflow of the wide-body dryer design. When a project was initiated to investigate means of producing agglomerated powder on a tall-form dryer, the hybrid tall-form was conceived. The hybrid design would combine the tall-form chamber with the high swirl airflow of the wide-body dryer and attempt to duplicate the high product quality of the wide-body dryer. This concept was counterintuitive, as one would not expect to be able to confine the swirling airflow within the relatively narrow tall-form chamber without

excessive wall deposits. Furthermore, this new dryer did not exist in commercial or pilot scale, so the approach of CFD modeling was chosen to determine if the tall-form hybrid dryer was plausible.

### **Discussion of the CFD Method:**

The CFD method has been available for decades, but only in the last 10 to 15 years has it really been practical for efficiently solving industrial problems. The clear reason for this increased usefulness is the ever-decreasing cost of high-speed computers and memory. As little as 10 years ago we found model sizes in the tens of thousands cells to be the practical upper limit and this limit was due more to solution time considerations than model development time considerations. With the increases in speed and memory, the limiting step is increasingly becoming CFD model (geometry and mesh generation) development time. As an example: a 4-processor SGI Origin with 1 gig of RAM is used to solve a 500,000 cell model, containing mass, momentum, energy, Reynolds Stress turbulence, particle tracking, species exchange, and heat conduction equations in only a few days. However, the setup of the geometry and mesh for this might model take a couple of weeks.

At its core, CFD is the discretization and solution of the Navier-Stokes partial differential equations. These fundamental equations of fluid mechanics describe the mass, momentum, and energy flow of a fluid subject to various initial and boundary conditions. While the process engineer does not really need to fully understand these equations to derive utility from the CFD method, for those requiring more detail White [1] provides a good undergraduate discussion, while Currie [2] offers a more advanced treatment of the Navier-Stokes equations. Like all the really good PDE's, closed form solutions for the Navier-Stokes equations have only been found for the most simple of cases. It is therefore necessary to use some sort of numerical technique to solve complex industrial problems. CFD is one such technique.

The commercially available (as opposed to a university, proprietary or special purpose research code) FLUENT CFD code was used for the modeling discussed in this article. FLUENT solves the Navier-Stokes equations using a finite volume based method. Using the FLUENT code the quantities of mass, momentum, energy, and species are conserved throughout the computational domain. The computational domain boundaries in the spray dryer problem are the physical walls of the dryer along with

the various inlet and outlet openings. Each boundary is subject to mass, momentum, heat, and species transfer. The solution is an iterative one where the velocity field for the current iterative step is computed based on the pressure field from the previous iterative step. The mass continuity equation is used to develop a pressure correction term for the current iterative step. Based on these pressure correction terms the entire pressure field is updated. Using this revised pressure field the velocity field is recalculated. The iteration continues until the pressure and velocity corrections are reduced below some specified value for all points in the computational domain. The turbulence, energy, species, and material-properties equations are updated with each of the pressure and velocity iterations. As is the case with the pressure and velocity calculations, the iteration continues until the corrections to the turbulence, energy, species, and material-properties equations is reduced below a specified value.

### **Cases Modeled:**

It was discussed briefly above that changes were being considered for a tall-form powdered milk spray dryer. The existing dryer operates efficiently generating a non-agglomerated product. Consumer testing showed

that an agglomerated product is more desirable so a project was initiated to develop a process to agglomerate the product within the tall-form dryer. Agglomeration within a spray dryer is accomplished when the primary liquid feed is brought into contact with a portion of the returned powder fines from the dryer's outlet cyclones. As mentioned previously, the proper combination of product moisture, temperature and particle concentrations are required to achieve a good agglomerated product without creating heat induced product defects. Specifically, the process engineer is concerned with preventing protein denaturation and the formation of insoluble particles in the dried power. Initial processing attempts with the tall-form dryer resulted in good agglomeration, but unacceptable amounts of protein denaturation. In contrast, experiments performed with the high-swirl wide-body dryer produced excellent agglomerated powder without the protein defects. These results led to an investigation into the differences between the two dryers with major focus on the temperature, humidity, and flow patterns of the atomizing zones.

The goals for the CFD modeling effort were twofold. The first goal was to demonstrate that the technique would accurately model a process (e.g., the processes occurring in the baseline wide-body and tall-form dryers) whose

operating parameters and process variations were known. This step is critical to the modeling effort both in terms of calibrating the baseline models and to provide insight into how the model outputs compare to physical effects in the actual dryer. What, specifically, do we mean by “...provide insight into how the model outputs compare to physical effects in the actual dryer”? For example, the current state of the art in commercial CFD codes does not allow one to predict chamber fouling. It is true that the models can capture the impact and reflection of a particle with a solid wall, but there is no "canned" mechanism that allows one to specify that a particle with a given temperature, mass, and water content will or will not stick to the dryer surface causing deposits. Another effect not easily modeled by current commercial codes is the mechanism of particle agglomeration. In this process, the wet feed mixes with returned dry powder in the top of the dryer and agglomerates into larger particles. Although significant, these modeling limitations were not serious limitations to the overall project. The process engineers, skilled in the art of spray drying, were able to make conclusions about dryer wall fouling and particle agglomeration based on their knowledge of particle concentration, particle water content, particle temperature and drying air conditions within the chamber. This ability to

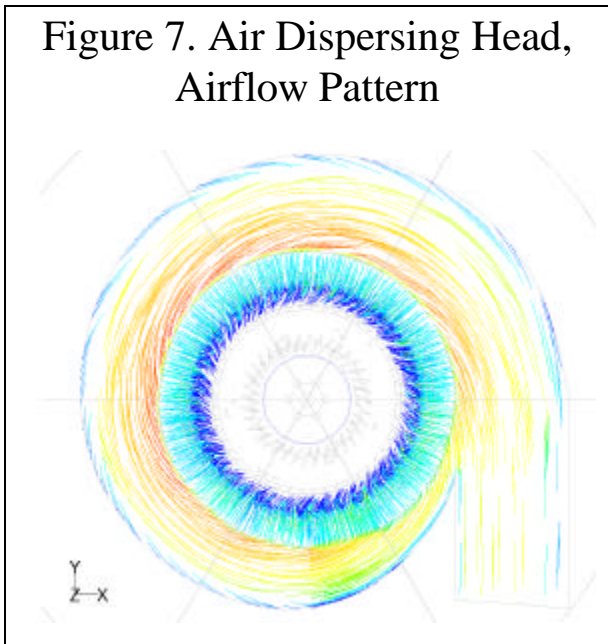
correctly interpret the CFD results and make conclusions about non-modeled events proved essential to the overall success of the project. Clearly, CFD modeling provided the process engineer with data he could not get elsewhere.

The second goal of the CFD modeling effort was to help the process engineer discover the correct geometry and process parameters necessary to make the fines-return agglomeration process work well in the tall-form dryer. In the proposed tall-form modification the inlet scroll and air disperser design from the wide-body dryer would be used on the tall-form chamber. Once the baseline model of the wide-body dryer was validated one could be sure that the airflow through the scroll and air disperser was correct. Therefore, one could use the same geometry on the tall-form dryer and have good confidence in the CFD results for the proposed dryer modification.

The validation process for the two baseline CFD models consists of comparing the CFD results to the actual dryer in as many ways as possible. Examples of the comparisons include mass flow rate (both fluid and solid particles), temperature, humidity, pressure drops, and particle concentrations. Clearly, one of the benefits CFD offers to the process engineer is the ability to "look inside" the dryer and see what's going

on. Without a massive and impractical instrumentation effort, one cannot know all of the above variables at all points within the dryer. However, at certain points like inlets and outlets it is practical to measure the actual conditions. One concludes, therefore, that if the physical dryer and the CFD model are given the same inlet conditions and the CFD model predicts the same outlet conditions as are measured on the actual dryer, then the CFD model is doing a good job of correctly modeling the internal behavior of the dryer.

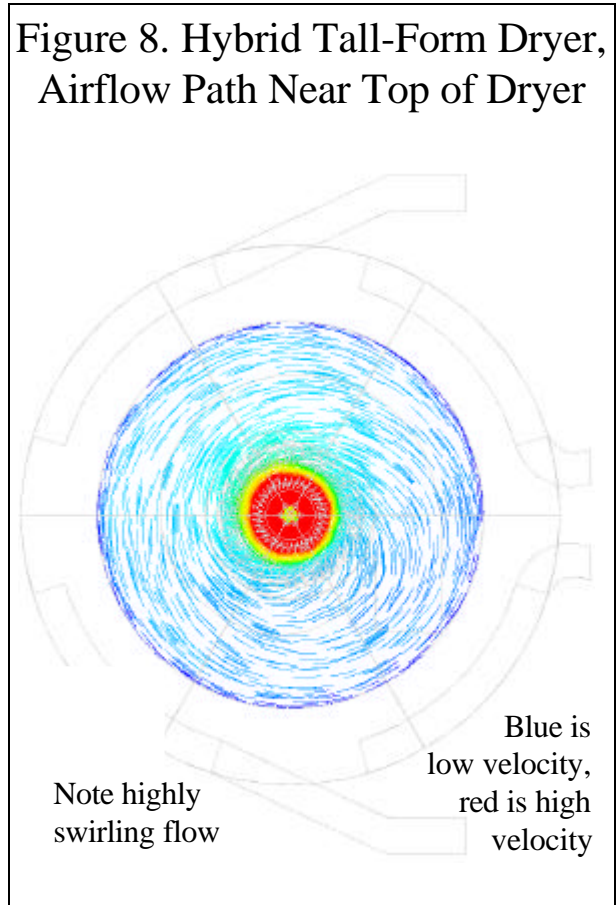
Figure 7. Air Dispersing Head, Airflow Pattern



As is the case with any scale-up problem, the true measure of success with the CFD technique would be whether or not the results allow one to make good design decisions.

With these goals in mind, both the tall-form and wide-body dryers were modeled with differing combinations of axial and swirling airflow.

Figure 8. Hybrid Tall-Form Dryer, Airflow Path Near Top of Dryer



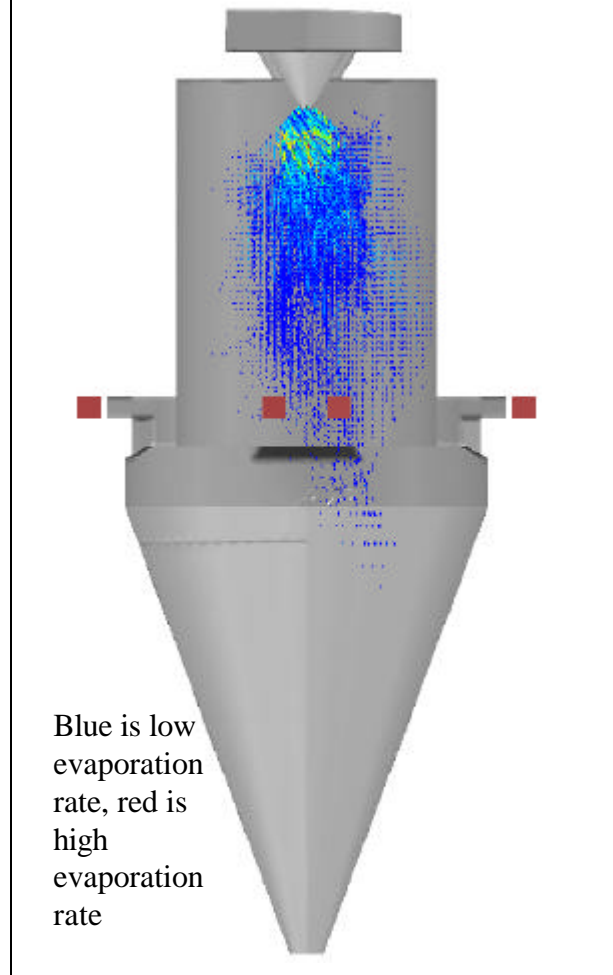
Probably the greatest operating concern when the project started was that the relatively narrow tall-form dryer chamber would be incompatible with a highly swirling airflow because of the high likelihood of centrifuging the powdered-product onto the dryer walls. Recognizing the swirl rate was a key variable; the CFD models were built to allow one to quickly change the angles on the air directing vanes that generate the swirling flow. Figure

7 shows the swirling flow being developed in the air dispersing head and Figure 8 shows the developed flow further into the dryer.

Recalling that geometry changes are the most time consuming aspect of a CFD model, it was desirable to be able to change the vane angles without having to make a geometry change to the model. This functionality was accomplished by modeling the vanes in the air disperser as a porous media, rather than as discrete vanes. The porous media model in Fluent allows one to specify flow resistance as a combination of Darcy's Law term and an inertial loss term. For the turbulent flow conditions modeled, only the inertial loss term, of the form  $\Delta P = 0.5\rho V^2 k = q * k$ , is required. (In the preceding equation “ $\Delta P$ ” is the pressure drop through the porous media, “ $q$ ” is the dynamic pressure defined as  $q = 0.5\rho V^2$ , “ $\rho$ ” is the air density, “ $V$ ” is the air velocity, and “ $k$ ” is an experimentally determined constant relating the pressure drop to the dynamic pressure.) The “ $k$ 's” (loss factors) are specified to capture the infinite flow resistance perpendicular to the vane surface and the relatively low resistance in the air gap between adjacent vanes. Comparing the porous media results to a test case, where the vanes were modeled explicitly, validated this approach. With the porous media vane approximation it is easy to change the

vane angles with a few keystrokes in the input file.

Figure 9. Hybrid Tall-Form Dryer, Product Evaporation



This approach saved an immense amount of modeling time when compared to the alternative of generating new vane geometry for each vane angle to be analyzed. Several combinations of vane angles were tried in order to determine a setting that appeared unlikely to develop deposits on the dryer wall or

ceiling. Along with changing vane settings, various feed nozzle combinations and nozzle lance angle settings were examined. These manipulations were performed to see how a change in the location of the primary feed spray affected the overall particle tracks within the dryer. The CFD modeling of the existing tall-form and wide-body dryers indicated it was necessary for the agglomeration to occur in an area that satisfied specific conditions for particle concentration, temperature, moisture and drying air humidity. By changing the vane angles on the air disperser along with the nozzle spray patterns, it was possible to create nearly the same conditions within the agglomerating zone of tall-form hybrid dryer as are present within the wide-body dryer. Furthermore, by examining the particle trajectories, temperatures and drying air humidity contours, one could reasonably well deduce where wall fouling was likely to occur. Figure 9 shows the location of the evaporating product cloud within the dryer for the optimized vane settings.

At the conclusion of the modeling we had developed a set of vane angle and spray nozzle settings to evaluate during field-testing. In the actual order of things, we performed tests on the wide-body dryer first to evaluate how well the agglomeration zone and process could be moved and controlled. Seeing that the CFD results

correlated very well with the wide-body dryer field tests, we went back to the CFD to determine the necessary settings to be tested in the hybrid tall-form dryer. Following these models, a commercial tall-form dryer was modified and tested. The next section will discuss the results of both the wide-body and hybrid tall-form field trials.

### **Field Testing:**

Plots from the first model of the hybrid tall-form dryer, utilizing the same guide vane settings and spray nozzle configuration as in the wide-body dryer, showed a very large area of high humidity extending to within half a meter from the chamber walls. This moisture distribution indicated a very high probability of chamber wall fouling. Additional combinations of nozzle position and reduced guide vane angle (to reduce swirl velocity) were modeled until a reasonable compromise configuration was identified. This configuration showed that one should be able to run the hybrid tall-form dryer without fouling and achieve a good agglomerated product. While this CFD data was useful, the consequences of making a mistake on the actual hardware dictated that we develop more confidence that the CFD results are accurate reflections of the real process. To develop this confidence

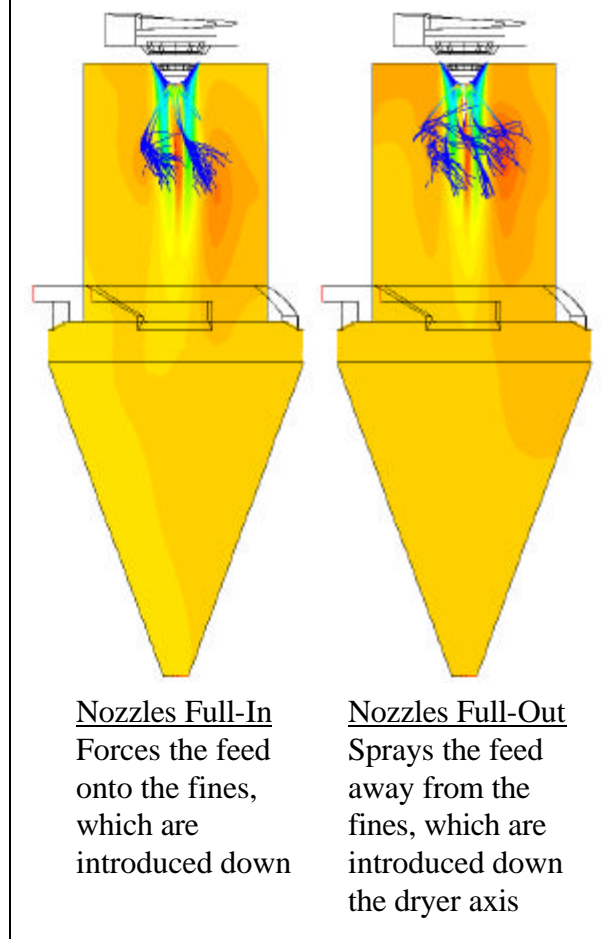
we chose several test points in the wide-body dryer to validate the CFD results against actual dryer performance.

Reconfiguring the wide-body dryer for the trials was a relatively simple procedure requiring only the adjustment of the guide vane assemblies. The model predicted that the agglomeration zone would become hotter, denser and extend deeper within the drying chamber as a result of reducing the swirl angle. This was proven to be true as the bottom of chamber (cone) fouled quickly with moist powder and the degree of agglomeration increased significantly. The presence of protein denaturation confirmed that the temperature within the agglomerating zone had increased significantly. These results were as anticipated. However, we were surprised when powder deposits formed on the upper sidewalls of the chamber. These were not anticipated, as the humidity along the walls was not excessive. Through additional querying of the CFD model, it was confirmed that the wall deposits were not caused by high ambient humidity within the dryer. Instead, the deposits were the result of some medium size droplets from the primary liquid feed carrying through to the wall by inertial force before they could completely dry. This effect was predicted by the CFD, but we had not specifically looked for it in the CFD results before

we observed it during the field test. The observation of droplets penetrating to the dryer wall raised our level of concern for the feasibility of hybrid design because the drying chamber was, after all, narrower than in the wide-body dryer.

With the accuracy of the CFD modeling technique proven during the wide-body dryer trial, focus shifted toward more detailed analysis of the proposed hybrid tall-form configuration. The modeling efforts took on two primary goals. First, we needed to identify what could be done to lower the concentration of primary spray and returned fines particles in the agglomeration zone to reduce the degree of agglomeration within the confined space of the narrower tall-form dryer. And second, the chamber air patterns, particle trajectories and particle moisture as they impacted wall had to be compared with the wide-body dryer results to determine if the concerns about increased chamber fouling in the hybrid were real. The model results indicated rotating the nozzles away from the returned fines stream and reducing the amount of fines returned to the atomizer could control the particle concentrations within the agglomerating zone.

Figure 10. Hybrid Tall-Form  
Dryer, Effect of Turning  
Product Feed Nozzles



This effect is shown in Figure 10 if one observes that the “full-out” nozzle pattern directs the product away from the injected fines, which are introduced from the center of the air dispersing head. We also found that because of the narrow chamber diameter, there was an upward flow of dry air at the chamber wall that functioned as a partial barrier to reduce the direct impingement of wet

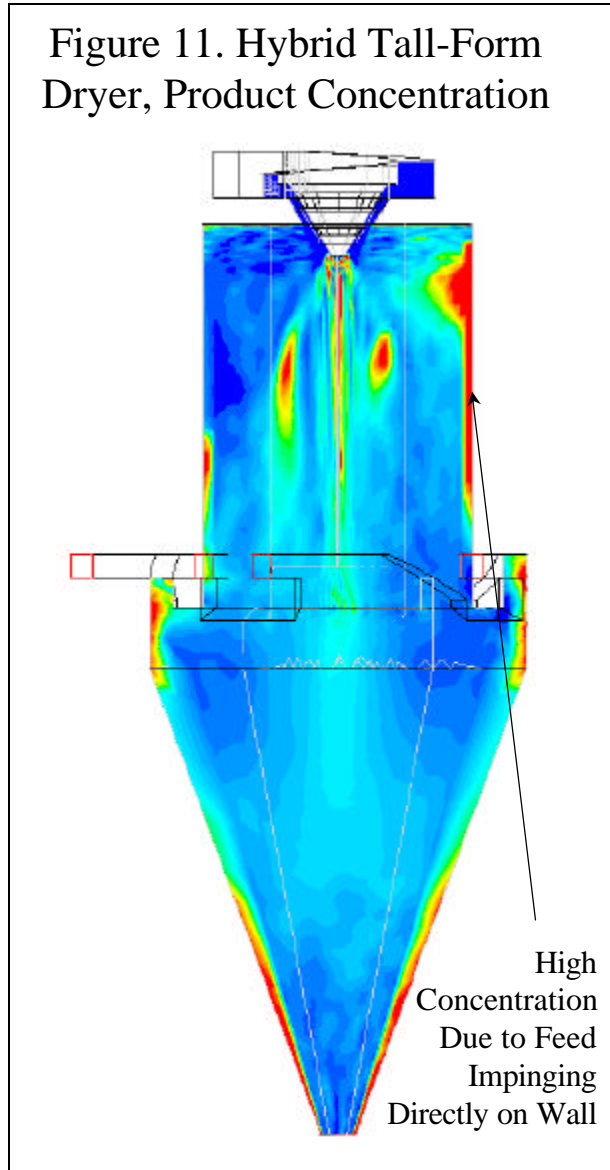
primary particles, from the atomizer nozzles, against the dryer wall. These findings gave the project team enough confidence to commission the design and fabrication of a test air disperser and atomizer assembly and schedule a field trial of the hybrid concept on a commercial tall-form dryer.

Before running any product into the hybrid tall-form dryer, a number of water trials were completed to observe the spray patterns and look for any obvious problems with the test configurations (i.e. water droplets accumulating on the dryer roof or walls). As predicted by the model, the rate of swirl of the drying air had a dramatic effect on the characteristics of the atomization cloud. Specifically, increasing the swirl velocity raised the cloud in the chamber, shortened its overall length and reduced the time the individual nozzle sprays were identifiable (a measure of spray cohesiveness). Conversely, as the swirl velocity was decreased, the atomization cloud pushed deeper into the chamber, became narrower, more wide-body, and the individual sprays were more discernable. At the end of the water trials, it was determined that the configuration tested on the wide-body dryer would be the starting point for the product runs.

The test strategy for all of the product trials was to start with a minimum agglomeration configuration and then

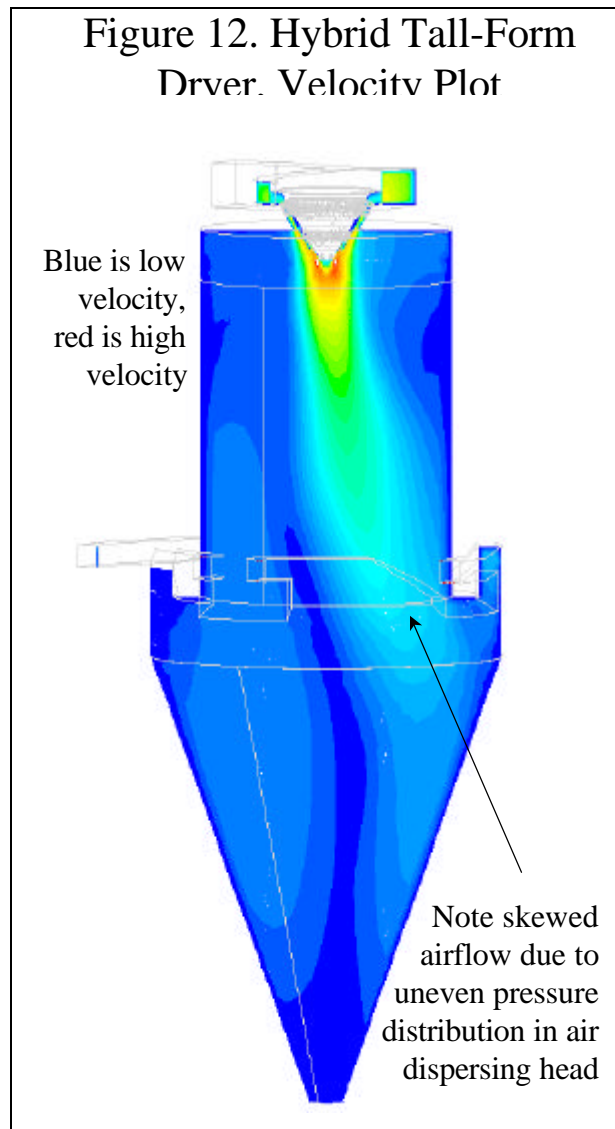
gradually adjust the dryer parameters until the desired agglomerate was achieved.

Figure 11. Hybrid Tall-Form Dryer, Product Concentration



In the first test, the product characteristics responded to the parameter changes as expected and were comparable to those from the wide-body dryer trial, including the protein denaturation.

Figure 12. Hybrid Tall-Form Dryer, Velocity Plot



The chamber fouling was light, although it must be mentioned that the run was only of 4-5 hours duration, and asymmetrical. This asymmetry is of particular interest as it was predicted by the CFD model and later found to be the result of non-uniform flow distribution within the inlet air scroll. The high particle concentrations along the wall are shown in Figure 11. The asymmetric

axial flow was noted on several velocity plots, one of which is shown in Figure 12. Steps are being taken to make the inlet flow more uniform.

For the second trial, the swirl velocity was increased to more rapidly mix and disperse the primary and fines particles. The reasoning behind this change being that shortening the contact time and reducing the particle concentrations in the agglomeration zone leads to a reduction in protein denaturation. The configuration for the second trial moved the processing conditions to more closely simulate the successful air disperser and nozzle configuration used in the wide-body dryer. The results of this second trial were very good. The protein denaturation was significantly reduced and good agglomerated product was achieved. There did not appear to be any additional chamber fouling although this was difficult to access because the material from the first trial was still present on the chamber walls.

Based on the results of these two trials, an extended run is planned to optimize the product qualities and access the long term effects of the high swirl air flow on dryer chamber build up and run time between cleaning cycles. Additional modeling has already been completed to correct the pressure distribution within the inlet

scroll and balance the air mass rate into the air disperser.

### **Conclusions:**

The decreasing cost of high speed computers coupled with the evolution of increasingly powerful software has made it practical and cost effective to solve complex industrial problems using computational fluid dynamic (CFD) modeling. In the specific case discussed in this paper, a new, non-intuitive, approach to spray drying was developed to produce a more desirable, higher value, agglomerated milk powder.

Clearly, CFD modeling of complex systems is not an inexpensive proposition. The total cost to develop the spray dryer models and run the multiple cases for this project exceeded \$100M. Lessons learned in the modeling will allow this cost to be reduced in subsequent models. However, when one considers that this cost can be measured in days or, in the case of high value products, hours of operating profit from an average capacity commercial spray drying plant, CFD modeling to assure process modifications and improvements work as designed, the first time, is a bargain.

---

[1] White, Frank M., **Fluid Mechanics**, McGraw-Hill, Inc., New York, 1979.

[2] Currie, I. G., **Fundamental Mechanics of Fluids**, McGraw-Hill, New York, 1993.