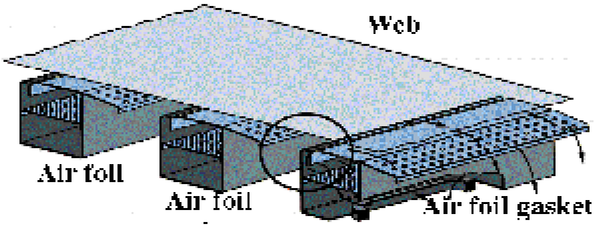
Computational Fluid Dynamics Helps Solve Web Stability Problem in Industrial Dryer

By Sheng-Yi Lee
W.R. Grace & Company
Columbia, Maryland



A web stability problem in an air flotation dryer was solved by using computational fluid dynamics (CFD) to analyze the airflow and determine the effect of changing the nozzle geometry.

The analysis showed the sensitivity of the air cushion pressure to the contour geometry of the foil top, including the location and opening of the slot. This made it relatively simple to solve the immediate difficulty without reducing web speed. The CFD analysis made it possible to evaluate many more geometrical variations than would have been practical with experimental testing alone and also provided airflow vectors throughout the dryer which could not be measured in the laboratory.

Air flotation dryers are used in many industrial web applications where idler rollers cannot be used because of the need to avoid contact with the web until it has dried. Air foil flotation nozzles, sometimes referred to as Bernoulli type nozzles, provide single-sided web transport while avoiding disturbing the coating with high convection rates from pressure pad nozzles used on both sides of the web such as Coanda air bars. However, air foils do not provide as wide an operating range with respect to web tensionand slot pressure as do air bars. Web instabilities such as flutter or billowing may occur when airfoils are applied or operated outside their performance envelope.

The conventional method for determining the operating envelope of a given dryer and web combination is laboratory testing. The primary purpose of such testing is to determine the air foil cushion pressure which provides considerable information about the aerodynamics of a nozzle with a simulated web. The nozzle to be tested is positioned and fastened on a header connected to an adjustable air source. A plastic plate simulating a web is positioned at the proper height and level over the nozzle surface. A wall tap in the plate is connected to an inclined manometer to measure local cushion pressure. The air source is adjusted to provide the proper slot pressure for the test. The slot pressure, supply pressure and metering orifice differential pressure are measured and recorded. Wall tap pressure readings are taken on 1/8 inch increments over a representative nozzle pitch distance.

Two significant deviations from actual flotation conditions are made with this apparatus. First, the plate is not flexible like the web and thus does not assume the slight sinusoidal shape observed in airfoil flotation. Second, the plate is set at a fixed clearance and angle to the air foil surface and does not position itself such that the positive and negative pressure regions balance with gravitational forces on the web. Despite these facts, pilot dryer flotation results give information as to the appropriate simulation clearance of the experimental setup. Even though flotation does not take place, the quantitative results are useful as an analytical tool when related to actual flotation results.

Recently, Grace has begun performing CFD analysis on the air foil system. A key advantage of CFD is that it makes it possible to evaluate geometric changes with much less time and expense than would be involved in laboratory testing. A second advantage is that CFD provides far more detailed output information including, particularly, airflow vectors throughout the system which provide engineers with a much better understanding of why a proposed design performs the way it does. We selected FLUENT CFD software from Fluent Inc., Lebanon, New Hampshire, because the user is completely shielded from learning the intricate algorithms that actually are used in the analysis, making it unnecessary to become an expert in the underlying mathematics of the CFD process. Another advantage is that the program provides immediate graphical feedback during grid generation, model creation, and the solution process. Model development involves the following steps: 1) inputting the geometry 2) discretizing the flow domain 3) choosing the physical models 4) assigning boundary conditions and fluid property values 5) numerically solving the model equations and 6) postprocessing the numerical solution.

Various curvatures and slopes of the air foil contour are key geometrical features that are thought to be critical for determining the air flow and cushion pressure. If conventional Cartesian coordinates were used, the curves and sloped lines could only be approximated as "stair steps", thus the wall shear, which critically affects the air flow between the plastic plate and the air foil, could not be accurately modeled. In order to accurately simulate the important geometrical features, a special geometry feature of the FLUENT program, called body-fitted coordinates (BFC), was used. Body-fitted coordinates use a curvilinear coordinate system in which the grid lines conform to the geometry being modeled and need not be straight or orthogonal. The BFC grids, which discretize the flow domain, are topologically rectangular. In other words, each two dimensional fluid volume must have four sides, but each side does not have to be a straight line: the opposing sides need not be parallel and of the same length; the neighboring sides need not be perpendicular to each other. The entire simulation domain was discretized using a 105X40 BFC grid.

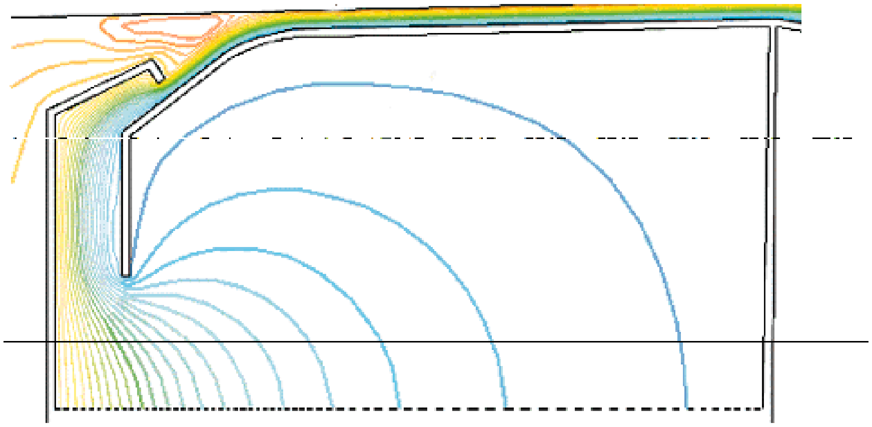
The mathematical model of air flow consisted of five partial differential equations: one for the conservation of mass, two for the conservation of momentum, one for turbulent kinetic energy and one for turbulent kinetic energy dissipation rate. The model assumes 2-D, steady-state, isothermal and incompressible flow. The partial differential equations were solved numerically using a finite volume method and the SIMPLE algorithm for coupling pressure with velocities.

The velocity vectors provided by the analysis results revealed that the air jet, after exiting out of the slot, splits into forward and backward streams. The backward stream causes an eddy beneath itself and above the slot. Occurring in a dryer, the backward stream would usually be in the countercurrent direction with respect to the web travel. The local mass balance about the slot disclosed that 10% of the air jet exiting out of the slot flows backward.

The backward stream, as it flows to the left side of the air foil, entrains air from below and expands in the cross section. At the left boundary, the volumetric flow rate of the backward stream has increased by 14%.The cushion air pressure calculated by the analysis agreed very well with the experimental results. The major transitions including the slight negative pressure before the pressure rise, the lowest negative pressure and the recovery to ambient pressure computed by FLUENT all matched the measurement nearly perfectly. The agreement in the pressure peak value is within 15%. However, the negative pressure section of the computed pressure curve did not drop as low as the experimental measurement, differing by up to 50%.

The analysis results showed that maximum air velocity occurs at the slot. After exiting out of the slot, the air velocity decreases and the kinetic energy converts to static pressure bringing about the cushion pressure rise. A slight negative pressure rise at 2 inches is caused by the small recirculation flow induced by the backward stream. At 2.4 inches, the pressure peak occurs at the point of impingement with the plastic plate. Downstream of the impingement point at 2.6 inches, the spacing between the plastic plate and the air foil top decreases. As the air velocity increases, the kinetic energy increases at the expense of static pressure. Hence, the reduction in pressure at 2.6 inches is caused by a combination of wall friction and the increase in velocity. From 2.8 to 6 inches, the clearance is constant. Here, wall friction is the dominant cause for the calculated pressure drop so the pressure decreases linearly at constant air velocity. Above the trailing edge at six inches, the clearance increases significantly with a corresponding decrease in the air velocity. The conversion of kinetic energy to pressure is greater than the loss due to wall friction, so the static pressure recovers from the lowest negative value until reaching the ambient pressure.

In order to help solve the application problem mentioned above, cushion pressure curves for three different clearance values were plotted. For 3/16 inch clearance, the cushion pressure curve did not show a positive pressure region. In actual web flotation, this would cause the web to be drawn closer to the air foil. On the other hand, with 1/16 inch clearance, the positive pressure region is larger than the negative pressure region. The net effect is to push the web away. Interpolation of the curves suggested a clearance condition with comparable positive and negative pressure regions exist. Since a proper balance of the positive and negative pressures is a necessary condition for web stability, the computational study implied an optimum clearance in



Flow about the nozzle top and web support area is visualized by the stream function.

The CFD analysis showed the sensitivity of the air cushion pressure to the contour geometry of the air foil top including the location and opening of the slot. It also provided valuable information on the impact of specific geometric changes. This made it possible to solve the problem in much less time than would have been possible with experimental methods alone. In addition, the CFD analysis provided far better understanding of the dryer performance than can be obtained in the laboratory. As a result, Grace made the decision to use CFD as a key application analysis and trouble-shooting tool. Future work will include computational treatment of mechanically flexible boundary conditions to simulate web positioning and contour, experimental flow visualization and measurement of heat transfer coefficients.