

Computer Simulation Instrumental in Development of Inward Burning Matrix Burner

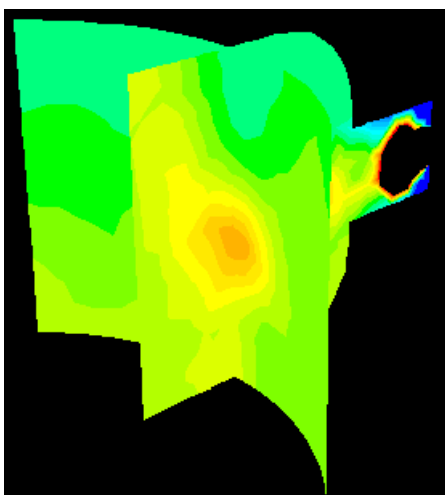


Fig 1. CFD simulation of air/fuel mixing in the burner plenum. The curved surface is the outside diameter of the burner matrix. The flat surface is a plane passing through the burner axis. An air/fuel inlet pipe can be seen on the right side of the figure. Average mass fraction of fuel is about 2.6%. A region of high fuel concentration, about 4.5%, can be seen directly opposite the air/fuel inlet pipe. Modifications to the the location of the fuel inlet reduced this to acceptable levels, see Figure 3.

Computer simulation was instrumental in the development of an inward burning matrix burner and heat pipe heat exchanger. The burner and exchanger are used to heat a Stirling engine on cloudy days when a solar dish, the normal source of heat, cannot be used. Geometrical requirements of the application forced the use of the inward burning approach, which presents difficulty in achieving a good flow distribution and air/fuel mixing. According to Mark S. Bohn, Principal Engineer, National Renewable Energy Laboratory (NREL) engineers solved the problem by using computational fluid dynamics (CFD) simulations to evaluate several inlet plenum configurations until they found one with just the right properties, which include good flow distribution and good air/fuel mixing with minimum residence time.

CFD simulations were also used to help design the primary heat exchanger needed for this application.

Dish/Stirling Systems engines generate electricity by focusing solar energy with a parabolic dish concentrator onto the heat-input end of a Stirling engine. This arrangement is of interest because it provides the potential for high solar-to-electrical conversion efficiencies and low costs can be achieved with mass production. Applications include utility scale generation, grid support, village electrification, remote power, end-of-grid power and irrigation pumping. Successful implementation of dish/Stirling systems requires hybrid operation (fossil-fuel co-firing) via reliable, efficient burner systems that can be installed at a capital cost increment competitive with conventional means for providing electricity from fossil fuels in a given market, e.g., diesel generator sets. Burner system lifetime is also a concern as are air emissions in many markets.

In most applications, a hybrid capability is needed to maintain operation at night or under cloudy conditions. In a hybrid installation, the thermal interface between the concentrated solar flux and the heat input end of the engine, a sodium heat pipe receiver, allows heat input from a burner as well as the parabolic solar dish concentrator. Metal matrix burners have received a great deal of attention lately because of their ability to burn fossil fuels with very low emissions of nitrogen oxides. In this type of burner, a premixed air/fuel stream flows through a porous metal matrix. A significant fraction of the heat of combustion is released as infrared radiation from the matrix. Because of this heat removal from the combustion zone, maximum temperatures are well

below the adiabatic flame temperature, resulting in emissions of nitrogen oxides (NO_x) as low as 10 ppm at 0% oxygen without exhaust gas recirculation.

Matrix burners have typically been used in an outward burning arrangement in which air and fuel mixing takes place inside a cylinder-shaped matrix and combustion occurs as the mixture flows outward through the matrix. In the dish/Stirling application, the configuration of the receiver demanded a cylindrical inward burning approach. This approach has been avoided in the past because of the difficulty of obtaining uniform flow in an annular shaped plenum.

NREL engineers decided that flow distributors could be used and still maintain a low pressure drop. The problem was that there are a large number of possible configurations. Using a build-and-test approach, it would have been expensive and time-consuming to test even a single configuration. Even after a proposed design was built, it would be very difficult to place instruments in the plenum and so engineers could only learn the temperature and flow conditions at a few points. This would make it impossible to measure the flow patterns in the plenum so engineers would be able to gain little understanding of why their design alternatives didn't work.

So, NREL engineers decided to use CFD to evaluate the inlet plenum and flow distributor configurations. A CFD analysis provides fluid velocity, pressure and temperature values at every point in the solution domain. This allows engineers to optimize fluid flow patterns or temperature distributions by adjusting either the system geometry or the boundary conditions, such as inlet velocity and temperature, wall heat flux, etc. Of particular benefit to this application is the fact that CFD enables detailed parametric studies that make it possible to optimize a design with relatively little time and expense.

Engineers selected FLUENT software from Fluent Incorporated, Lebanon, New Hampshire, to perform the analysis. This package was selected, first of all, because of its powerful meshing routines that automatically generate the mesh but give the user control over its density, skewness and the other parameters that determine its quality. FLUENT also offers a robust and well-validated combustion model.

NREL engineers used that model to determine if the preheated air/fuel mixture would preignite (burn upstream of the burner matrix). This was important because the dish/Stirling application requires recuperation of exhaust gas energy via combustion air preheat to achieve good thermal efficiency.

NREL engineers began by modeling the geometry of the plenum and air and fuel supply pipes. The CFD software read the basic geometry of the plenum from a Pro/Engineer model. The plenum forms a cylindrical annulus around the heat pipe and distributes the air and fuel from four inlet pipes to the outside of the burner matrix. With the geometry imported, engineers created the mesh and ran their

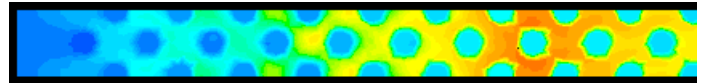
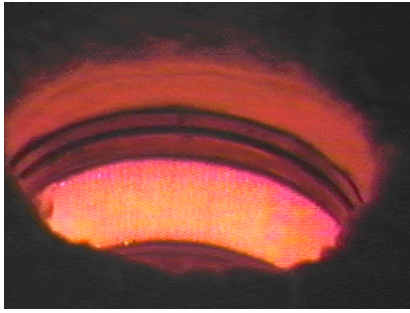


Fig 2. CFD simulation of the flow and heat transfer in the primary heat exchanger. Plane of visualization is normal to the pin axes and about halfway between the pin root and tip. Temperature contours are shown in the figure. Flow is nominally normal to the page for the first seven pins on the left. The flow then turns to the right and flows across the pins. The region of higher-temperature gas is near the point where the flow turns and the highest velocities are seen. Note that the pin temperatures do not exceed about 1250 K.

first analysis iteration. This process took about one day.

The initial iteration showed problems with air/fuel mixing in the plenum. [See Figure 1] Engineers then began considering different alternatives for the fuel injection location. In addition, flow distribution was less than ideal so this was also adjusted in the model. A key advantage of modeling the geometry with CFD is that from the very beginning engineers were able to get a complete picture of the flow patterns of each alternative they considered. These graphical results helped them to understand the effect of geometry and were instrumental in improving them. An example is the ability to visualize recirculation zones, which, through excessive residence times, can lead to preignition.

After solving the flow distribution problem, engineers turned their attention to the primary heat exchanger. This device transfers heat from the combustion gases to the heat pipe. The design requirements were 75 kW heat transfer to the heat pipe, 5000 Pascal pressure drop and 1150 K maximum metal temperature. Since preliminary

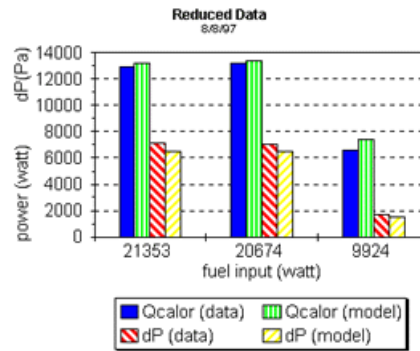


Photograph of the matrix burner operating with about 600°C preheated air/fuel mixture. View is from above the burner looking into the burner inside diameter at about a 45° angle. Note the uniformity of the burner radiosity.

calculations showed that enhanced heat transfer surfaces would be necessary, engineers began with working with a strip-fin configuration and later considered round pins. In both cases they evaluated a wide range of sizes and spacing of the fins and also considered both in-line and staggered placement for the pin-type fins. The analysis showed that strip fins provided inadequate heat transfer and would likely be subject to excessive thermal stress. Pin fins provided adequate heat transfer, acceptable pressure drop and were not expected to suffer from significant thermal stress. [See Figure 2] CFD was a critical element to the process because there are no literature correlations available for this complex geometry. (The combustion gas flow begins in a radial direction along the pin axes and then turns across the pin axes.) In addition, significant temperature change in the exhaust gas and resulting large property variations preclude the use of existing correlations. Finally, radiation heat transfer is an additional complicating factor that was relatively easily incorporated in the CFD work.

The cost and time involved in meeting their design objectives using the old build-and-test approach would have been so high as to make the project impractical. Using CFD, they were able to optimize their design in just a few weeks with zero hardware cost.

The next step was building a one-sixth scale prototype to validate their model. This system was designed and built in collaboration with engineers at Sandia National Laboratories, Albuquerque, NM, and



Comparison of CFD results with measured pressure drop and heat transfer. Pressure drop is that across the primary heat exchanger. Heat transfer is shown in the figure as the heat removed from the heat pipe by a calorimeter. Both were modeled with CFD for a range of burner throughputs and a correlation developed from those results. The abscissa is the higher heating value of the fuel delivered to the burner. The three runs are for air/fuel preheat temperatures of 470, 529, and 510°C.

provided the capability to test all features of the burner, primary heat exchanger, and heat pipe.

In this prototype system, only the heat pipe diameter was reduced to 1/6th full scale, all other dimensions were maintained at their full-scale values. The system was operated with the air/fuel mixture preheated to 550°C. [See Figure 3] Prototype test data showed very close correlation to the CFD results. Pressure drop across the primary heat exchanger and heat transfer through the primary heat exchanger agreed to within about 10% of the

CFD results. [See Figure 4] Maximum pin tip temperature, agreed to within 10C of the CFD model. In addition, the burner has been operated at the design air/fuel preheat temperature of 675C thus validating the CFD calculations used to predict preignition. These results have given the team confidence in the design, which is now undergoing full scale development.

This application clearly illustrates the advantages of using computer simulation to optimize burner and heat exchanger designs. The traditional trial and error approach involves high prototype building costs and gives engineers little insight into why their prototypes are or are not working. CFD simulation, on the other hand, provides a much higher level of information that shows them complete details on a design's performance and helps them rapidly iterate to a solution at minimal cost.