

Computer Simulation Helps Optimize Design of Innovative Rocket Motor

Chuck Margraves, Mechanical Engineer

Stone Engineering

Huntsville, Alabama

One of the most difficult challenges in pintle-controlled rocket design is configuring the nozzle so the gas pressure at the nozzle exit is equal to the outside air pressure in order to maximize thrust. Engineers at Stone Engineering Company (SEC) in Huntsville, Alabama, are reducing design time and cost by using computational fluid dynamics (CFD) rather than physical testing to determine the optimum configuration. CFD allows us to look inside our design to gain a far greater understanding than we were ever able to achieve with physical testing results alone in the past. The result is that we can see exactly where flow separation occurs for various nozzle geometries and fine-tune our design to maximize thrust under a wide range of flow conditions.

SEC provides technical support to the U.S. Army Missile Command and Space and Strategic Defense



Figure 1: Mach contours for boost position, with the pintle fully open

Command in the areas of propulsion and structures. The company has extensive "hands-on" experience in the analysis, design, development, and testing of solid, liquid, hybrid, and gel propulsion systems, metallic and composite structures, and gas generators. Extensive capabilities in structural analysis, ballistic prediction, combustion instability analysis, as well as an in-depth understanding of the requirements for today's systems place SEC in a position to move forward in our fields of expertise.

One of our most interesting current projects is a bipropellant gel rocket engine that uses an axial pintle to control the throat area and hence the motor thrust. The use of the movable pintle to control the throat area provides the potential to promote higher efficiency in the lower-thrust sustain phase of the motor burn, and provides a flexible response to the requirements of the application.

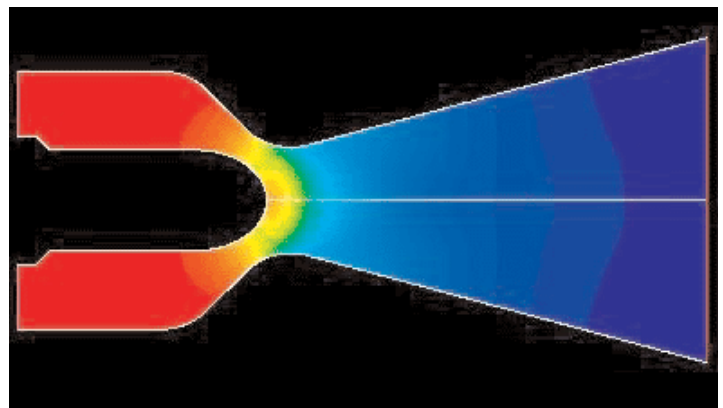


Figure 2: Pressure contours for boost position

Rocket motor design challenges

The pintle design helps overcome a basic challenge in rocket motor design. Combustion of the rocket propellant releases energy in the form of heat, which doesn't provide thrust directly, but instead raises the pressure of the gases. The nozzle has a carefully designed surface that changes the energy from gas pressure into velocity. At the nozzle entrance, the gases are at high pressure and low velocity while at the nozzle exit the gases have a low pressure and a high velocity. The nozzle expands along a carefully controlled contour so that the gases gain speed and lose pressure. The larger the nozzle cross-section at the exit, the faster the gases will be traveling as they exit. For optimum thrust, the gas pressure at the nozzle exit should be exactly equal to the outside air pressure. When the nozzle pressure is higher than the outside air pressure, not all the energy has been converted to exhaust velocity so energy is wasted. Alternatively, when the nozzle pressure is lower than the outside atmospheric pressure, the thrust differential causes drag that has a severely negative impact on performance.



Figure 3: Spring actuated pintle controlled motor

It is not difficult to optimize a rocket engine under fixed operating conditions. It is much tougher, however, to optimize the performance of an engine over a range of operating conditions. The geometry of the nozzle must be changed in order to keep the pressure of the combustion gases at the exit of the nozzle equal to or close to atmospheric pressure. In

the past, this has been done by using a complicated, heavy and expensive hydraulic control system to vary the throat area of the engine. SEC has developed a much simpler, lighter, and less expensive approach that replaces the hydraulics and control system with a spring that repositions the pintle in order to maintain a constant pressure in the combustion chamber. This innovation created the challenge of designing the spring and combustion chamber so that constant pressure can be maintained throughout a range of operating conditions.

Moving beyond the build and test approach

In the past, optimizing the spring and combustion chamber geometry would have required building multiple prototypes and testing each one in order to evaluate its performance as designers gradually iterated towards an effective design. One problem with this approach is that the prototypes are expensive to build and test. Another problem is that long lead-times are involved in both building and testing prototypes, which would have delayed the project by an indeterminable amount. In addition, there is the problem that only a limited amount of information can be generated during physical testing because of the cost and difficulty of instrumenting the combustion chamber. The basic operating parameters of the engine are not difficult to determine, but engineers would not be able to obtain detailed flow information from inside the combustion chamber so they would have to modify the design largely based on their intuition and experience, rather than on objective information.

We decided to use CFD on this project because we felt it could dramatically improve the design process by allowing us to explore different design alternatives without the cost and time involved in building and testing a prototype. We also felt that the ability of CFD to provide detailed information throughout the solution domain would speed the process of moving towards an optimized design. A CFD analysis provides fluid velocity and other variables for problems with complex geometries and boundary conditions. As part of the analysis, a researcher may change the geometry of the system or the boundary conditions and view the effect on the fluid flow patterns. CFD also can provide detailed parametric

studies that can significantly reduce the amount of experimentation necessary to develop new equipment or alter the design of existing equipment. It is therefore commonly used to reduce design cycle times and costs.

Simulating and optimizing the design

We began by building a model of the rocket engine using FLUENT, CFD software from Fluent Incorporated, Lebanon, New Hampshire. FLUENT substantially reduces the time required for analysis by largely automating the process of creating a high quality mesh. The easy-to-use pre-processor has a wide range of controls that make it straightforward to build a mesh and manage its quality. The two-dimensional, axisymmetric model we built had 254,000 elements, a 1500 psi pressure inlet boundary condition representing the fuel moving into the combustion chamber, and an ambient pressure outlet.

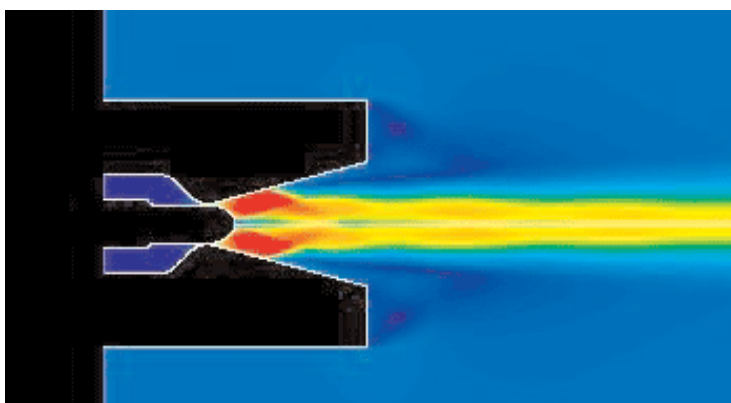


Figure 4: Mach contours for sustain position

In a rocket motor designed to operate at maximum efficiency at sea level, the gasses are allowed to expand from the chamber pressure to the atmospheric pressure at the nozzle exit in order to generate maximum thrust. This exit pressure is determined by the expansion ratio of the nozzle, or ratio of the area of the exit to the area of the throat. When the pintle is inserted, the ratio becomes very large so that the gases over-expand or under-expand to pressures either above or below atmospheric. At approximately 33% of the atmospheric pressure the gases no longer have enough momentum to continue to expand up the nozzle wall and instead, separate from it causing the thrust to be lost. Our goal is to try and force the gases to separate earlier, by

adding a step in the nozzle, so the loss will be minimized. CFD is helping us achieve this goal by providing complete visibility to flow patterns inside the nozzle and allowing us to determine the exact point at which separation occurs for a particular nozzle geometry.

Computer simulation in combination with physical testing has helped establish the basic design parameters for the motor. The initial analysis provided the forces acting on the pintle at various flow conditions. This information was used to develop specifications for the spring that controls the pintle. The analysis results showed good correlation to the initial test firings and provided far more detailed design information than we could ever obtain from physical testing. We ran the motor in the boost position with the pintle fully open and saw good correlation to the computer simulation. Then, we changed the simulation to move the motor into the sustain position. The simulation provided estimates of the mass flow rate and thrust which were used to calculate gas properties that were used as input to a thermochemical code. In the next phase of the design, we are going to run the simulation at steps in-between the boost and sustained positions in order to match the exit pressure to ambient pressure as closely as possible throughout the entire range of operation of the motor.

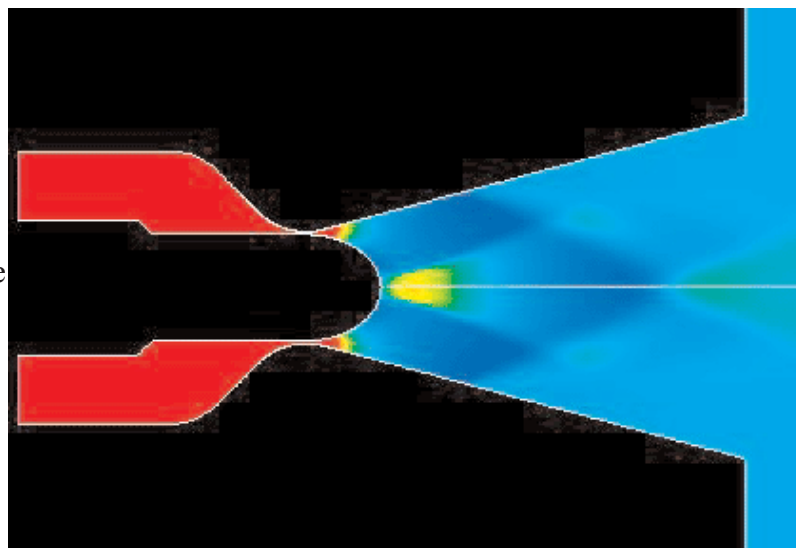


Figure 5: Pressure contours for sustain position