Conjugate compressible fluid flow and heat transfer in ducts

A computational approach to modeling transient, compressible fluid flow with heat transfer in long, narrow ducts is presented. The primary application of the model is for analyzing fluid flow and heat transfer in solid propellant rocket motor nozzle joints during motor start-up, but the approach is relevant to a wide range of analyses involving rapid pressurization and filling of ducts. Fluid flow is modeled through solution of the spatially one-dimensional, transient Euler equations. Source terms are included in the governing equations to account for the effects of wall friction and heat transfer. The equation solver is fully-implicit, thus providing greater flexibility than an explicit solver. This approach allows for resolution of pressure wave effects on the flow as well as for fast calculation of the steady-state solution when a quasi-steady approach is sufficient. Solution of the one-dimensional Euler equations with source terms significantly reduces computational run times compared to general purpose computational fluid dynamics packages solving the Navier-Stokes equations with resolved boundary layers. In addition, conjugate heat transfer is more readily implemented using the approach described in this paper than with most general purpose computational fluid dynamics packages. The compressible flow code has been integrated with a transient heat transfer solver to analyze heat transfer between the fluid and surrounding structure. Conjugate fluid flow and heat transfer solutions are presented. The author is unaware of any previous work available in the open literature which uses the same approach described in this paper.