

**CRUNCH Seminars at Brown, Division of Applied Mathematics**

**Friday - November 6, 2020**

**Introduction of CONVERGE CFD Software**

**By Dr. Daniel Lee, Convergent Science**

**Convergent Science specializes in modeling turbulent reacting flows using our CONVERGE CFD software. CONVERGE automatically generates a mesh at runtime, this eliminating all user meshing time even for moving boundary problems. CONVERGE v3.0 offers dramatic improvements in accuracy, robustness, parallel processing and speed for a wide range of applications including turbulence, rockets, batteries, medical flows, free surface flows, conjugate heat transfer, combustion, after-treatment, fluid structure interaction and gas turbine combustion modeling. In this presentation, an overview of the improved capabilities of CONVERGE v3.0 for a broad set of areas will be presented and the associated benefits to the end user outlined.**

**Use CONVERGE and never make a mesh again.**