CSE Engineering & CFD Modeling Services

Combustion Science & Engineering, Inc.

Computational Fluid Dynamics (CFD) use in Gas Turbine Combustor Design

2001 Adapco Users Conference

Andrew J. Hamer, P.E.

CFD is used to Improve Combustor System Design and Performance

Performance Parameters to be Considered:

- Flow Control
  - Where is the flow going and why?
  - How do I stop it from going where it wants to?
  - How do I enhance it?

- Pressure Drop
  - Need detailed understanding of loss distribution
  - – Sources and magnitude of each source
  - Difficult to determine from measurements

- Temperature Distribution
  - Where is it hot?
    - Will my part survive?
    - Will the downstream turbine survive?
    - Will it effect emissions?
  - Where is it not hot?
    - Will my part survive?
    - Will it effect emissions?

- Emissions Predictions
  - – Nox, UHC's, CO

- Pre-mixer development
  - Typically Fuel Peg/Swirler Combinations
    - Where should fuel holes go? How many? What size?
    - What should the vane shape be?
- Turndown
  - How low in load will combustor operate and maintain good emissions?
- Heat Transfer Coefficients
  - Map to a structural model for life prediction

- Operating Conditions
  - 1 to 20 atmospheres pressure
  - 300 K to 800K Inlet Temperatures
  - Natural Gas, Propane, Low-BTU, Jet-A
  - Models typically run at:
    - » base load
    - » full speed no load
    - » ignition conditions
  - Diffusion Mode, Mixed Combustion Mode, Pre-mixed Mode

- Using Medium Scale CFD Modeling for Guidance
  - Typical models
    - » ~ 1 to 2 million computational cells to obtain required mesh density
    - » Use Symmetry when possible
      - Typically 1/5 or 1/6 sector
    - » Very accurate geometric representation
    - » Higher order numerics to maximize accuracy

Physics:

Typically Chen variation of standard k-e for liners and transition pieces

- Better predictions of swirling and backward facing step flows

Non-linear 2-layer for swirler predictions

2-layer when heat transfer is required in a pre-mixer

Reaction Mechanisms

- Reduced mechanisms
- Public mechanisms – Westbrook & Dryer
- N-step now being tried by CSE

Physics:

Typically Working fluids simulated as:

- air (.2331% mass fraction of O_2 and 0.7669% N_2)
- methane (CH_4).

The products of combustion considered include:

- carbon monoxide (CO)
- carbon dioxide (CO_2)
- water vapor (H_2O)
- Nitrous oxides (Nox)

Physics:

All gaseous constituents are modeled as ideal gases

Specific heats of each species

- determined by polynomial fits as specified within the CHEMKIN Data Base

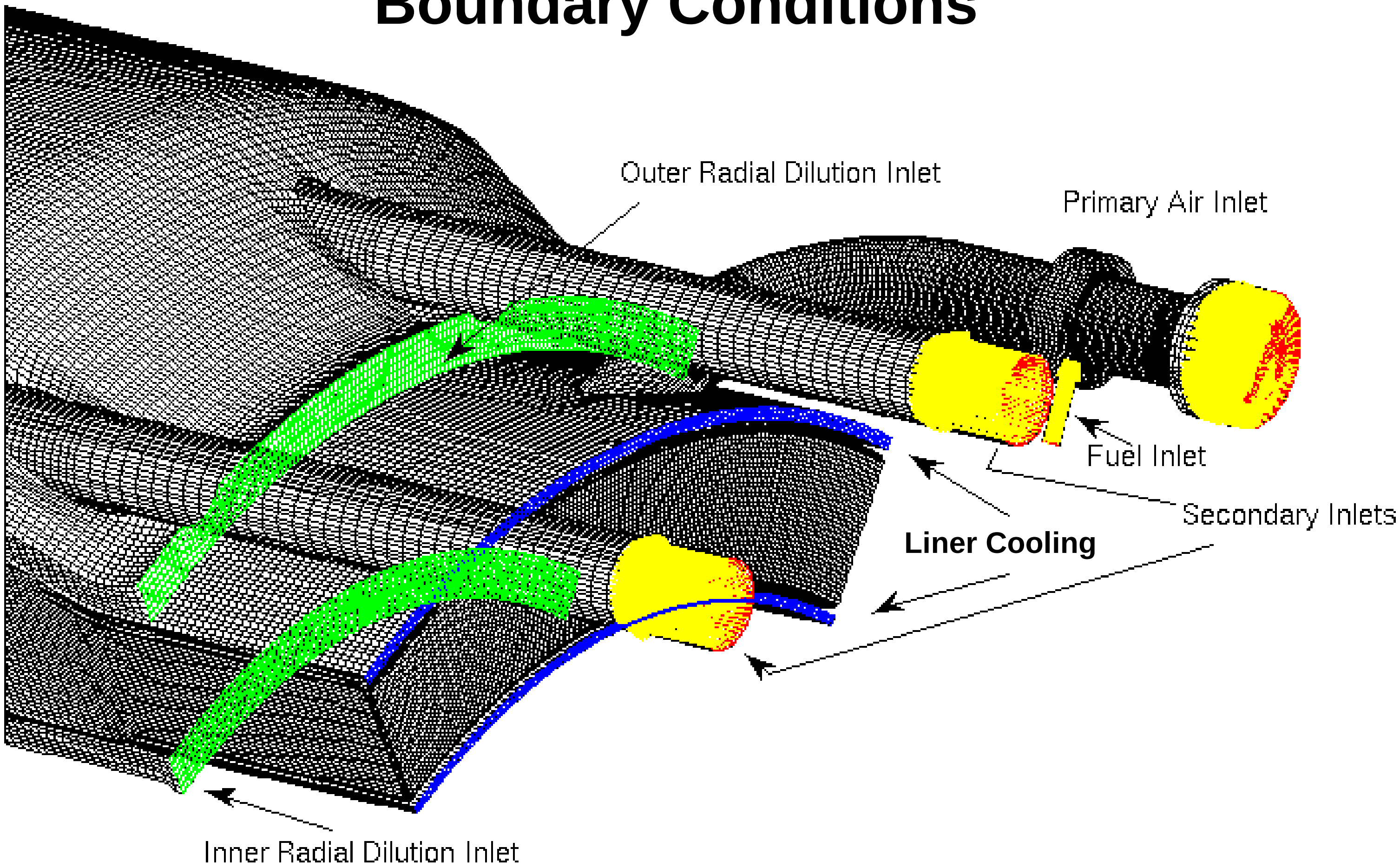
Molecular viscosity and thermal conductivity

- assigned to the bulk flow by means of mass weighting the contribution of each species present in the mixture to the respective parameters

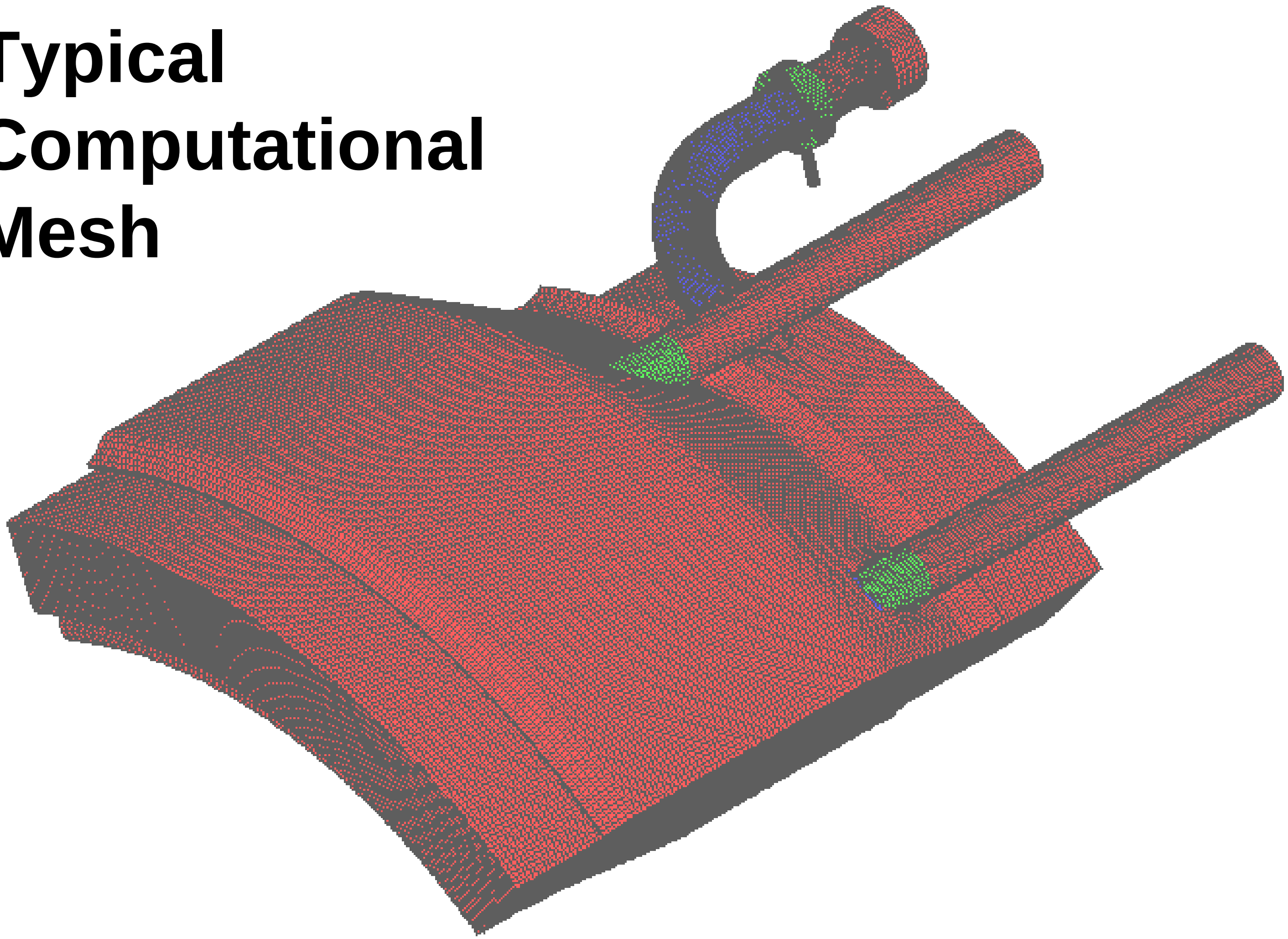
Project Goal:

- Understand the cause of a performance parameter difference between two similar designs
- Recover desired performance

Boundary Conditions



Typical Computational Mesh

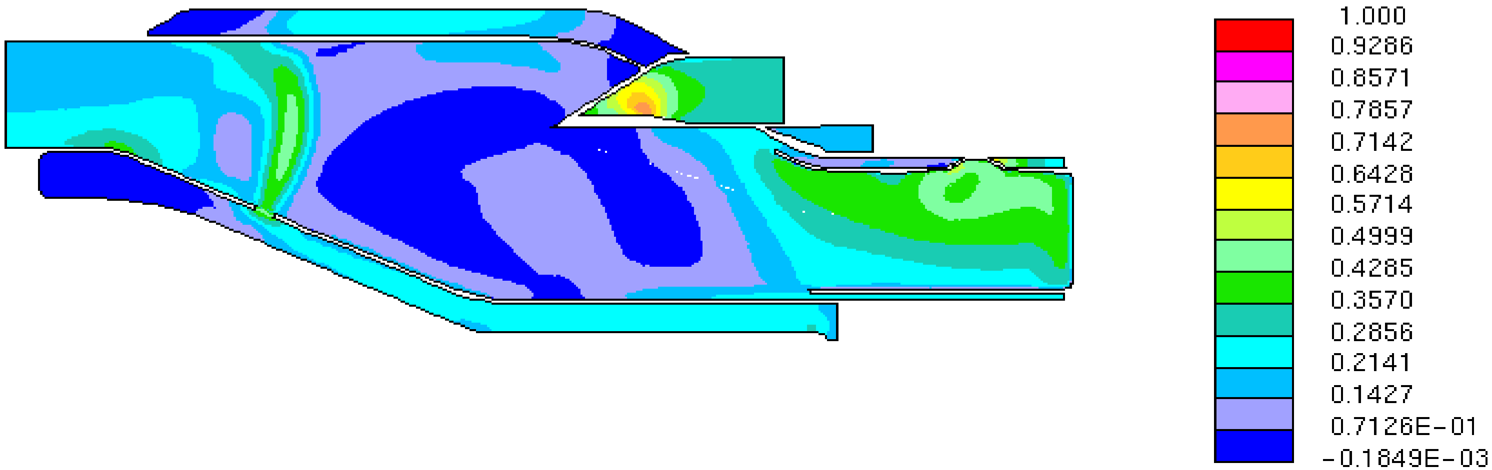


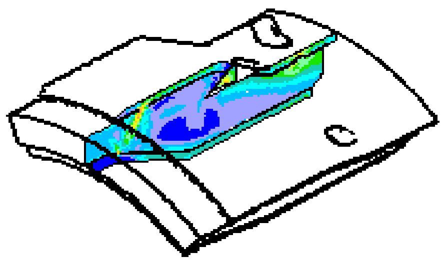
- Main Meshing Tool at CSE is
  - PROSTAR (Not PRO-AM)
- Why?
  - We do design
    - » Not a lot of CAD available
    - » Local Cell manipulation
      - Control on mesh distribution
      - Final Mesh is never the same as original
        - Always restructuring, refilling, re-projecting, re-sizing
      - Easy to “cut-in” geometric features
        - Macros, Panels, Scripts
      - Command Line is Powerful
      - Bullet-Proof – I know I am going to get there

CSE Engineering & CFD Modeling Services

STAR-CD
PROSTAR 3.10

Normalized Velocity



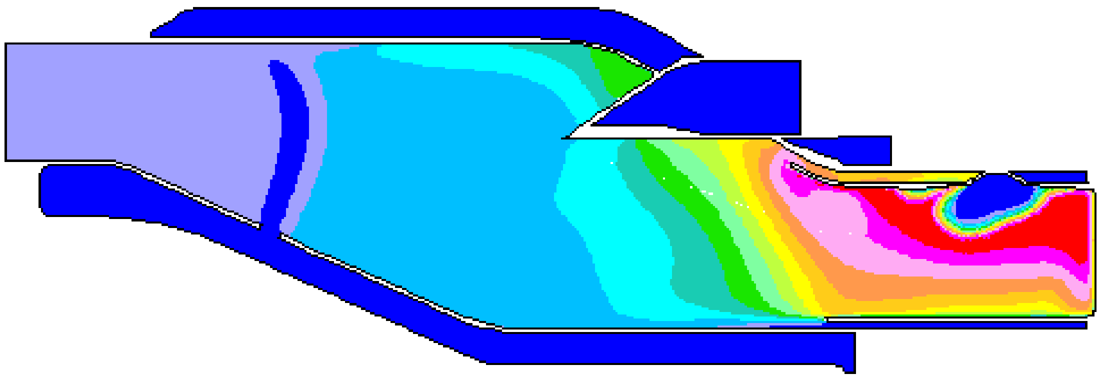


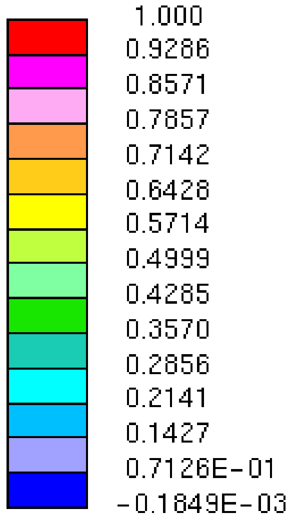
CSE Engineering & CFD Modeling Services

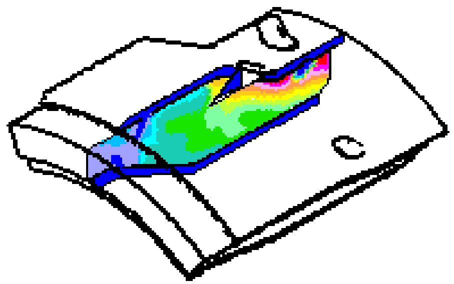
Combustion
Science & Engineering, Inc.

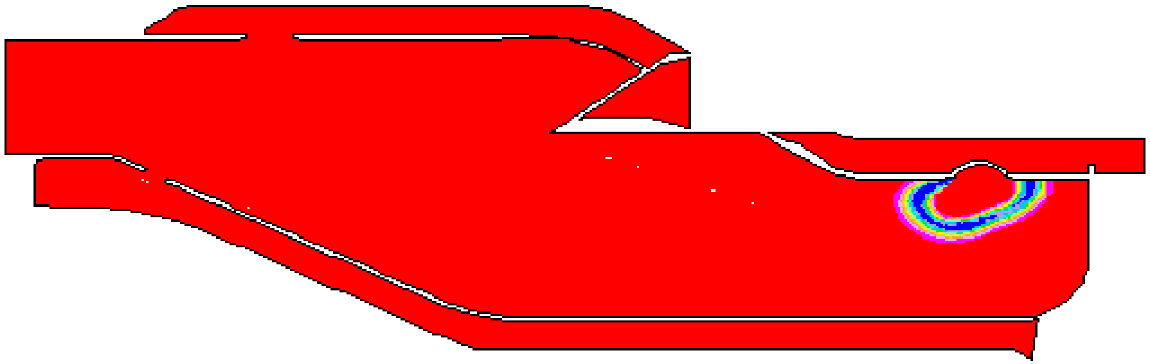
STAR-CD
PROSTAR 3.10

Normalized Temperature

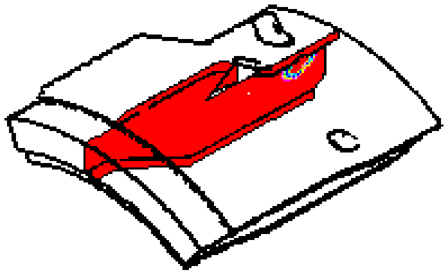








Reaction Rate

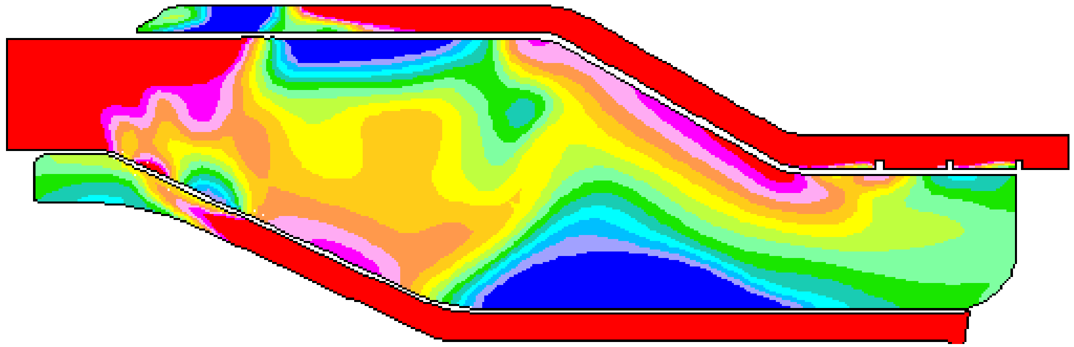


CSE Engineering & CFD Modeling Services

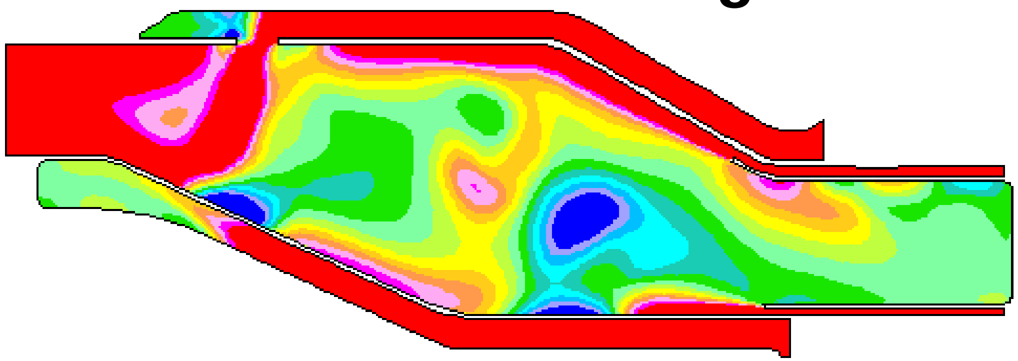
Combustion Science & Engineering, Inc.

STAR-CD

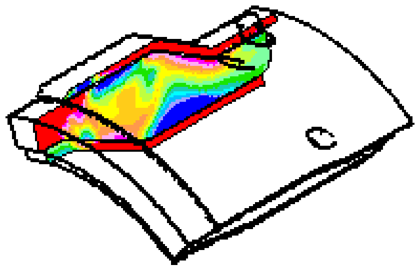
PROSTAR 3.10



Design B



Design A

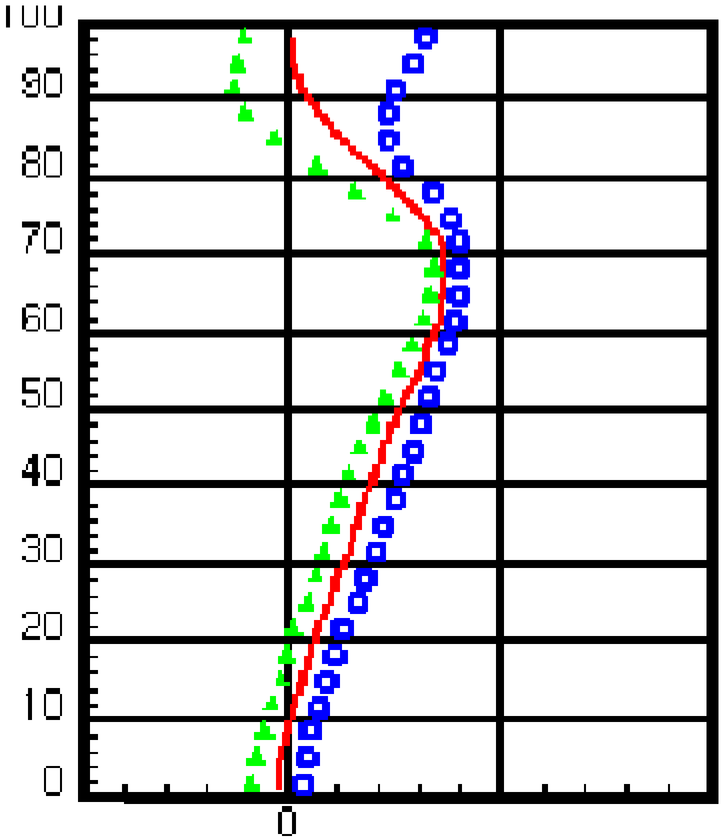


CSE Engineering & CFD Modeling Services

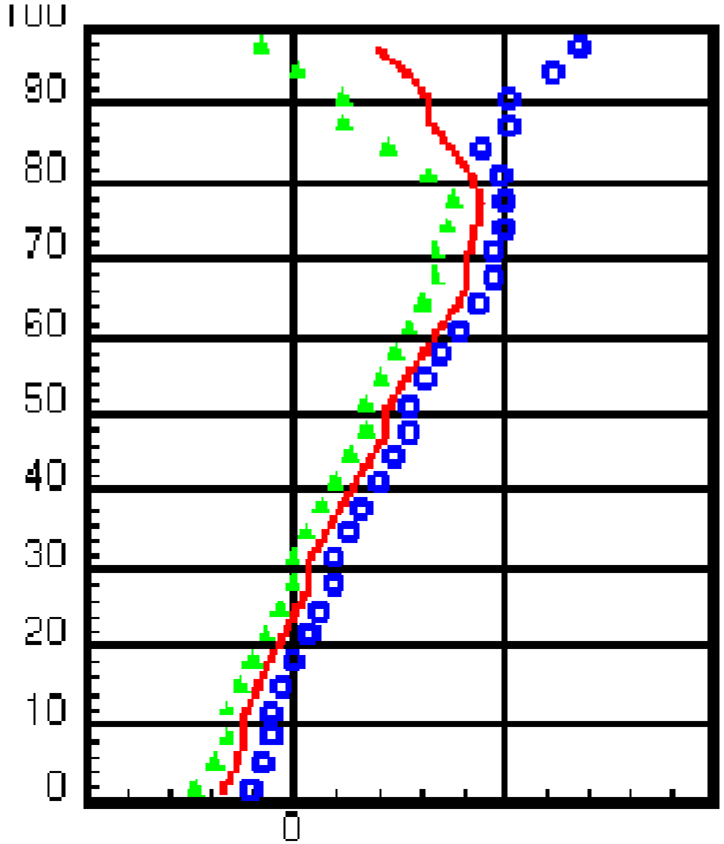
Combustion
Science & Engineering, Inc.

Axial Velocity versus Radial Position

1 Inch Downstream of Primary Tube



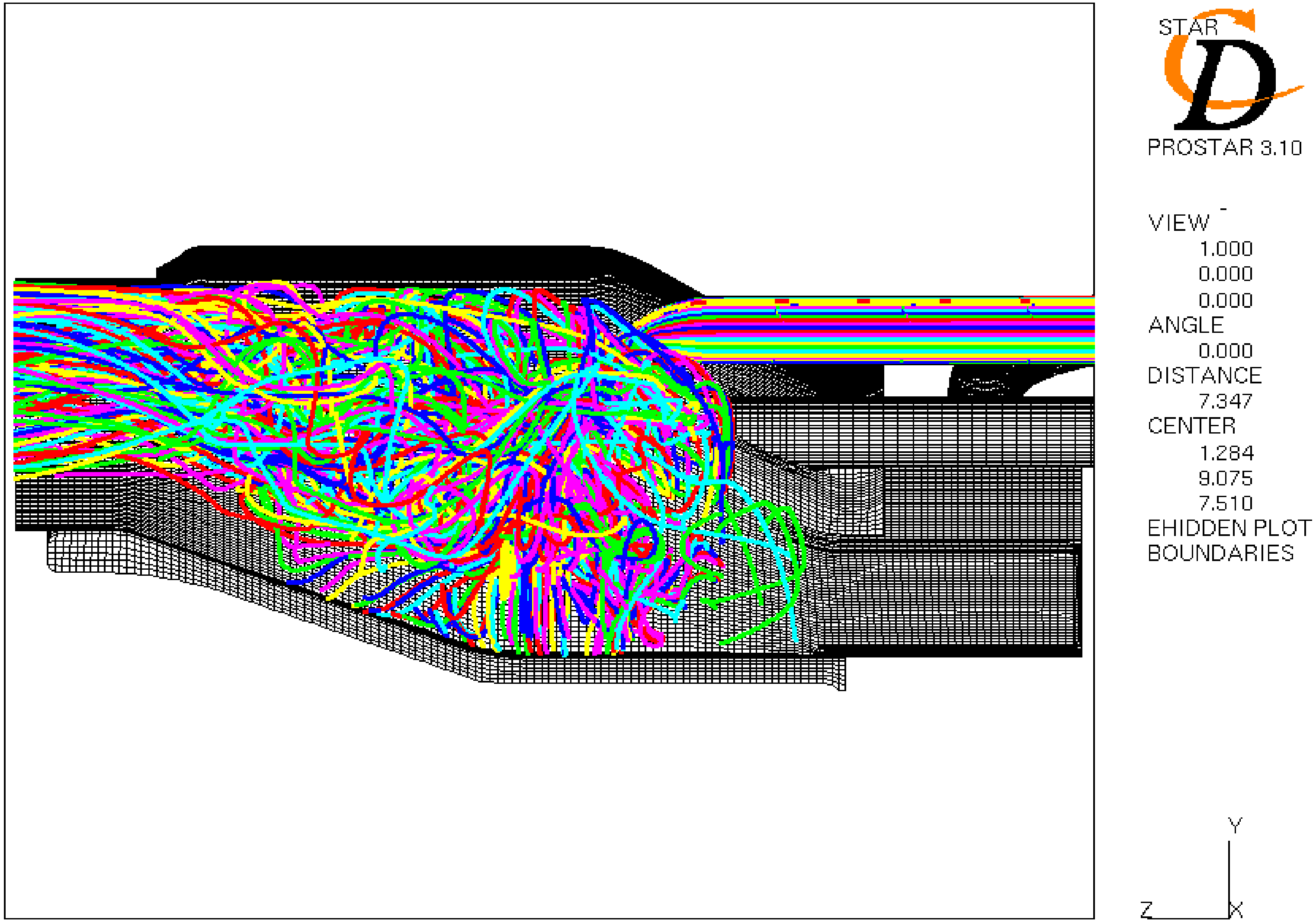
Design A



Design B

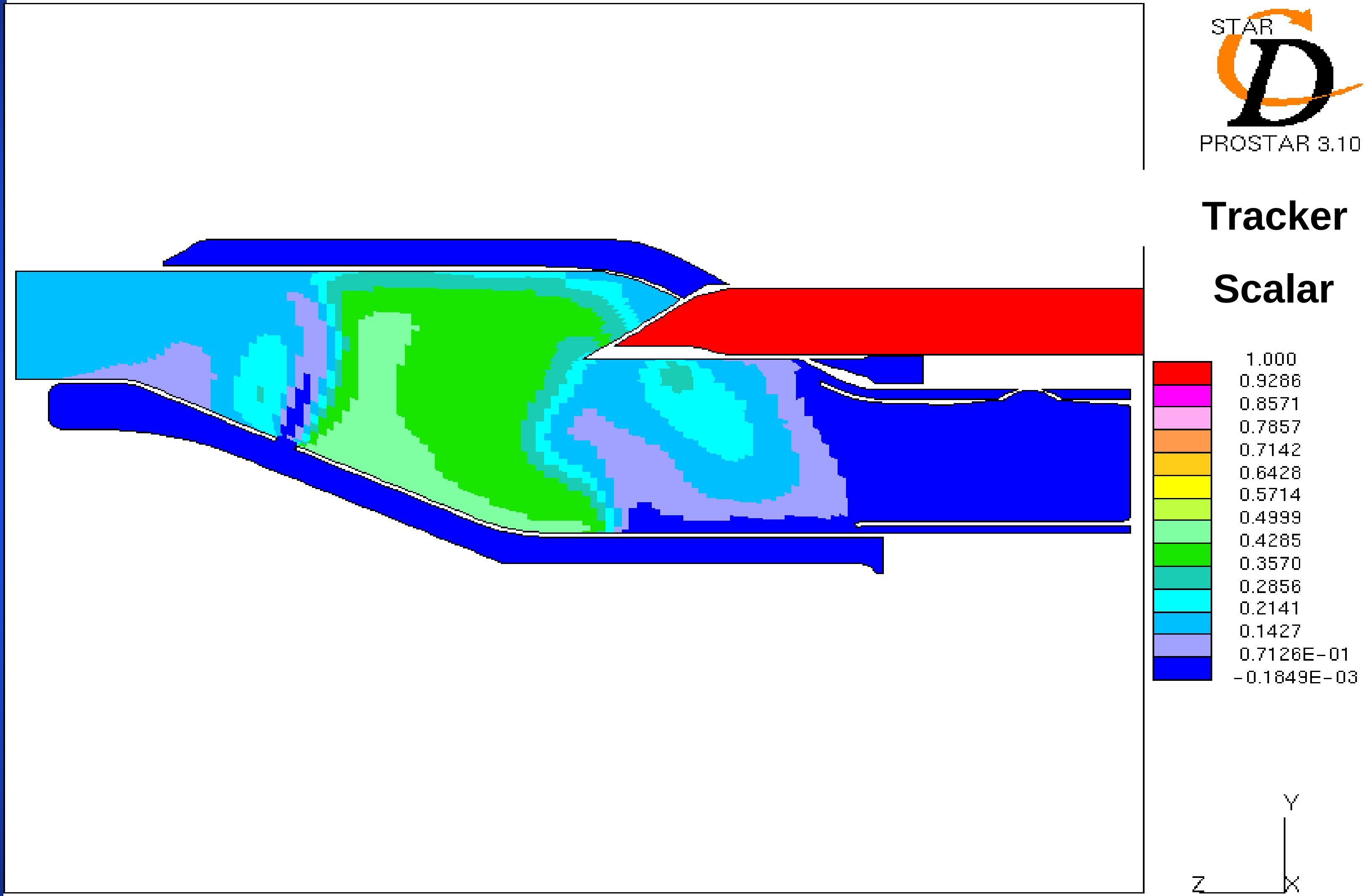
CSE Engineering & CFD Modeling Services

Combustion
Science & Engineering, Inc.

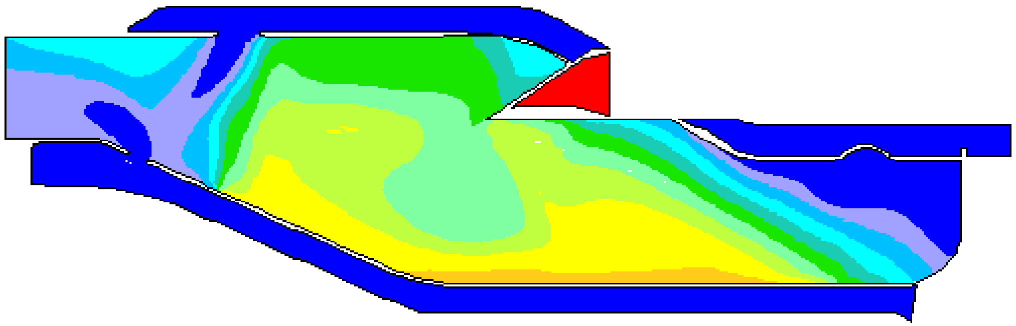


CSE Engineering & CFD Modeling Services

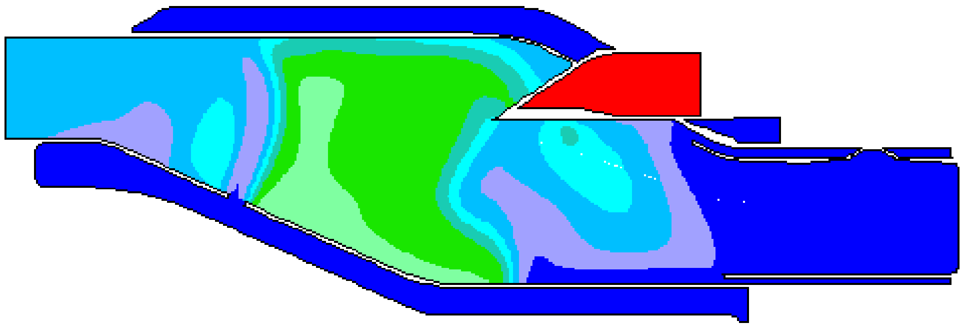
Combustion Science & Engineering, Inc.



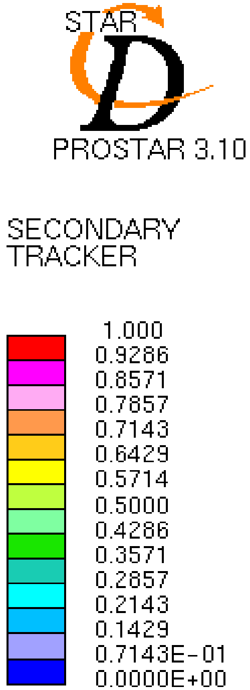
Baseline Tracker Movie

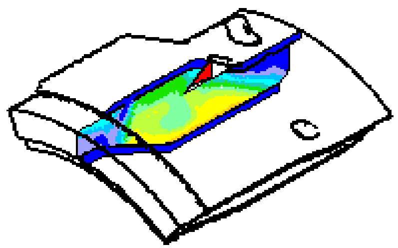


Design B

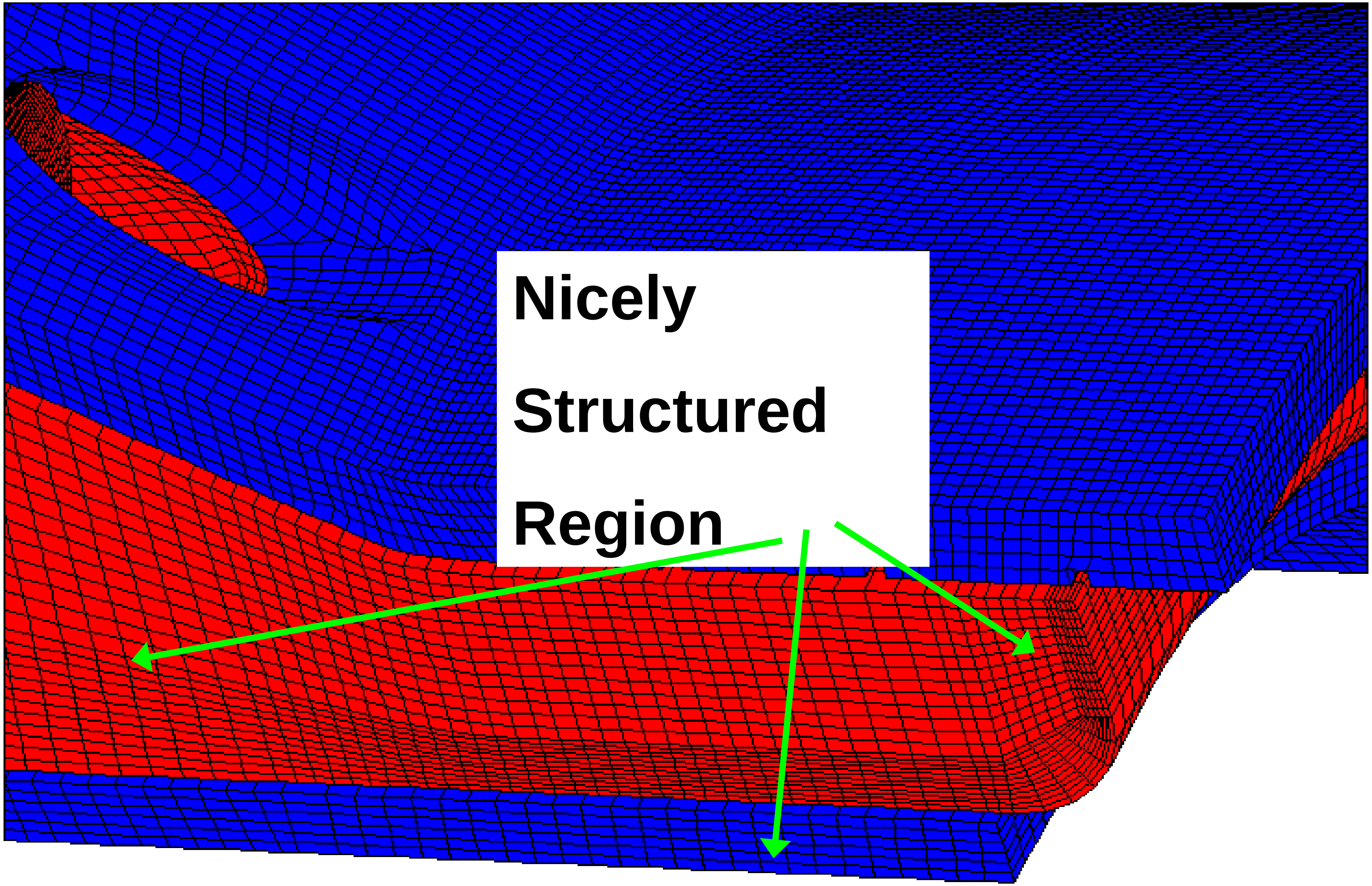


Design A



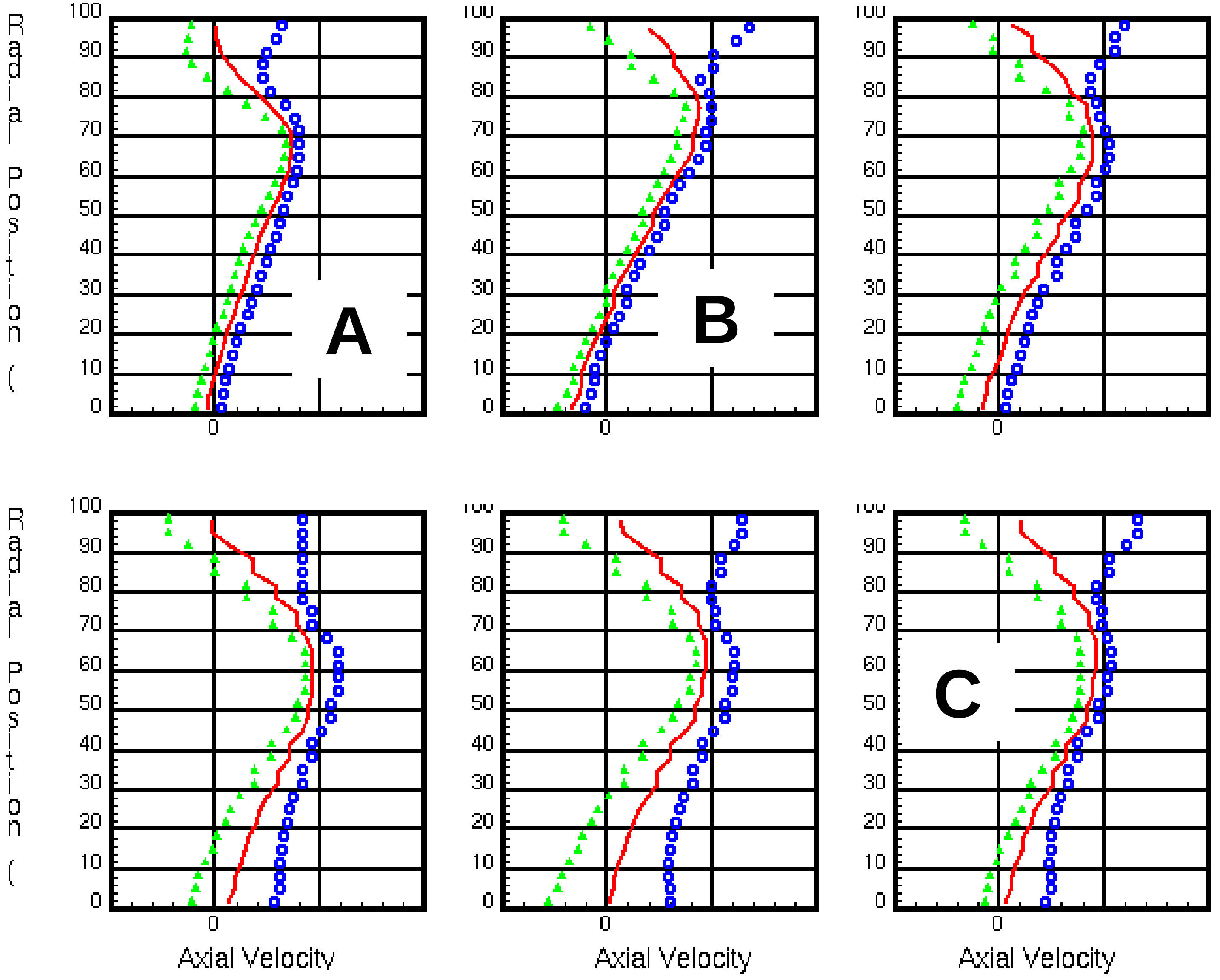


- How do we Prevent Secondary Flow from Entering the Primary Combustion Zone?
  - Develop concepts – “cartoons”
  - Eliminate Ideas based upon:
    - » Physics
    - » Correlations and rules of thumb
    - » Manufacturing – ability, cost



Nicely
Structured
Region

Combustion Science & Engineering, Inc.



Circumferential Average (Red), Maximum (Blue) and Mimimum (Green)

Modified Design Tracker Movie

CSE Engineering & CFD Modeling Services