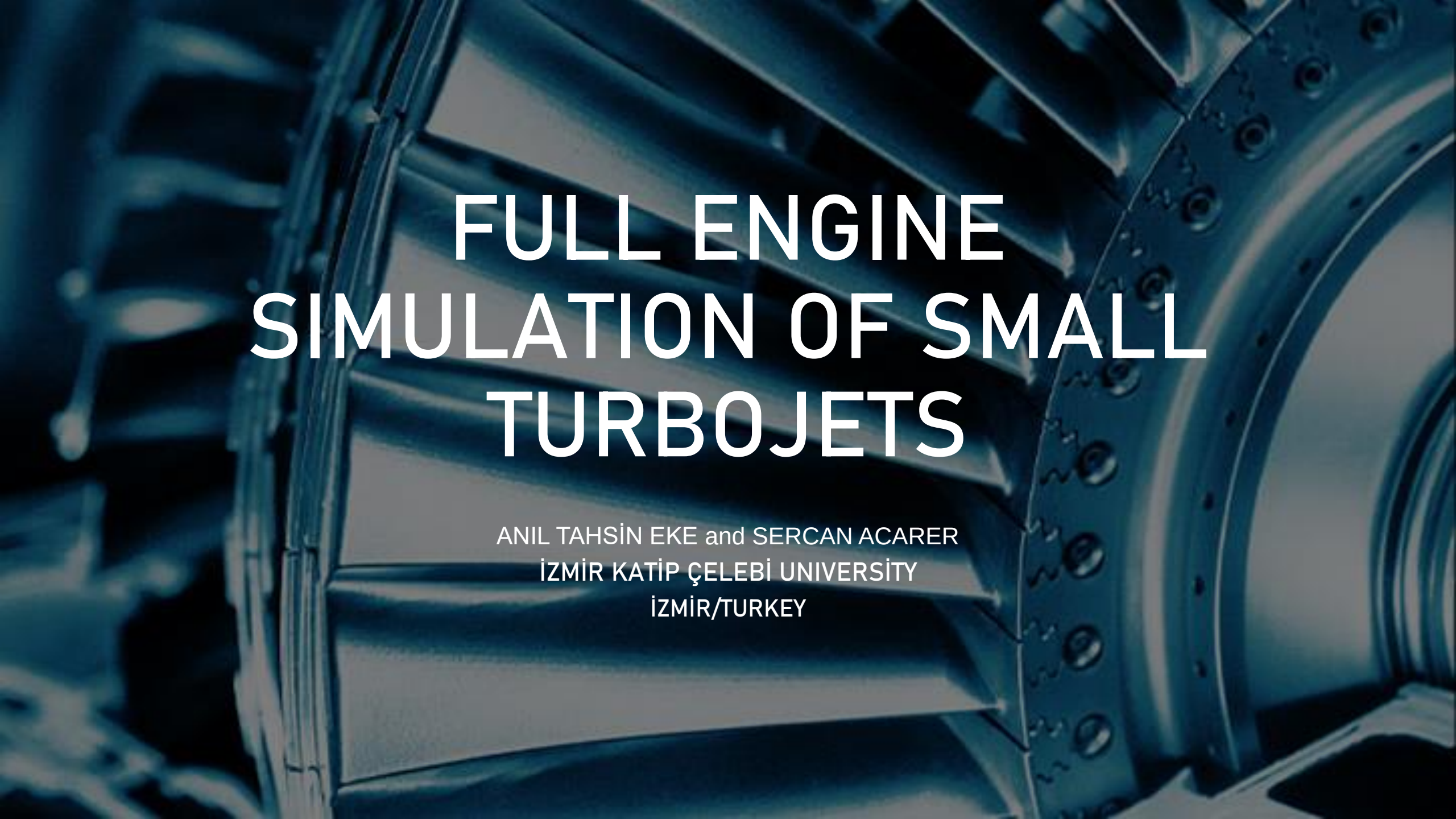


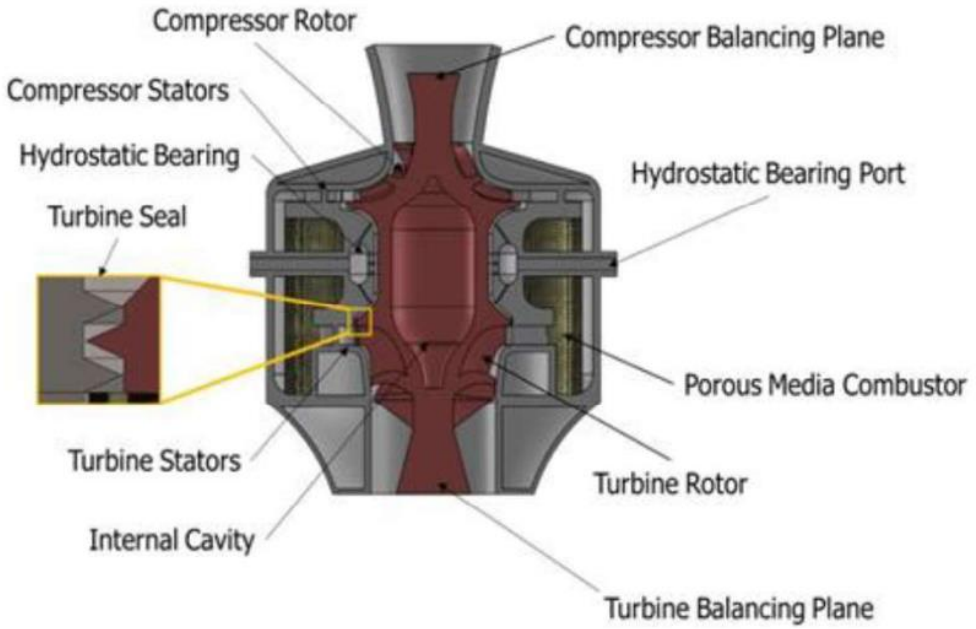
FULL ENGINE SIMULATION OF SMALL TURBOJETS

ANIL TAHSİN EKE and SERCAN ACARER
İZMİR KATİP ÇELEBİ UNIVERSİTY
İZMİR/TURKEY

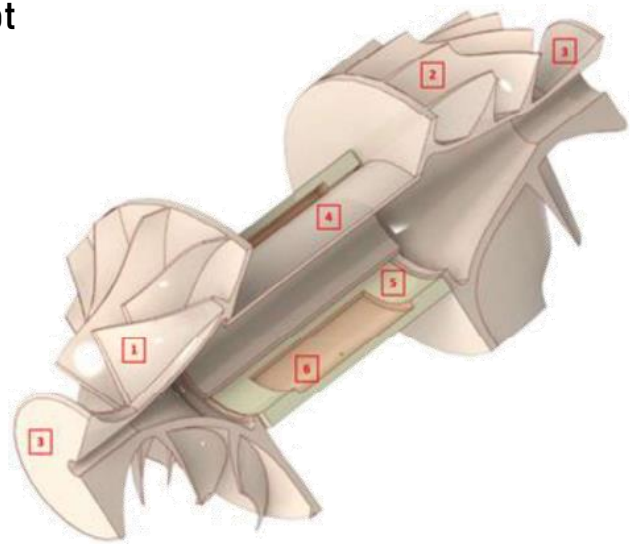
Case Background

- The case for this study is the gas turbine engine designed within the scope of NATO SPS G5939 Project.
- NATO SPS G5939 project aims to develop and demonstrate a micro gas turbine entirely produced using additive manufacturing. Moreover, the engine will be printed in its final assembly state, including a rotating part of the shaft with compressor and turbine, in a single uninterrupted print sequence.

Case Background



Additively printed engine concept



APE spool layout: (1) compressor, (2) turbine, (3) balancing masses, (4) connecting bearing shaft, (5) stationary bearing component, (6) pressurized supporting pocket

The Problem and Proposed Solution

- Component level analyses of the compressor, combustion chamber and turbine requires performance maps to be generated and their matching
- Coupled analyses, on the other hand, automatically ensures matching in a single simulation

The Problem and Proposed Solution

- Full engine approach requires only the compressor inlet and turbine outlet boundary conditions.
- The uncertainties on the interfaces are removed
- Thus, with a single analysis, detailed information about the flow field in the engine can be obtained at desired operating condition.

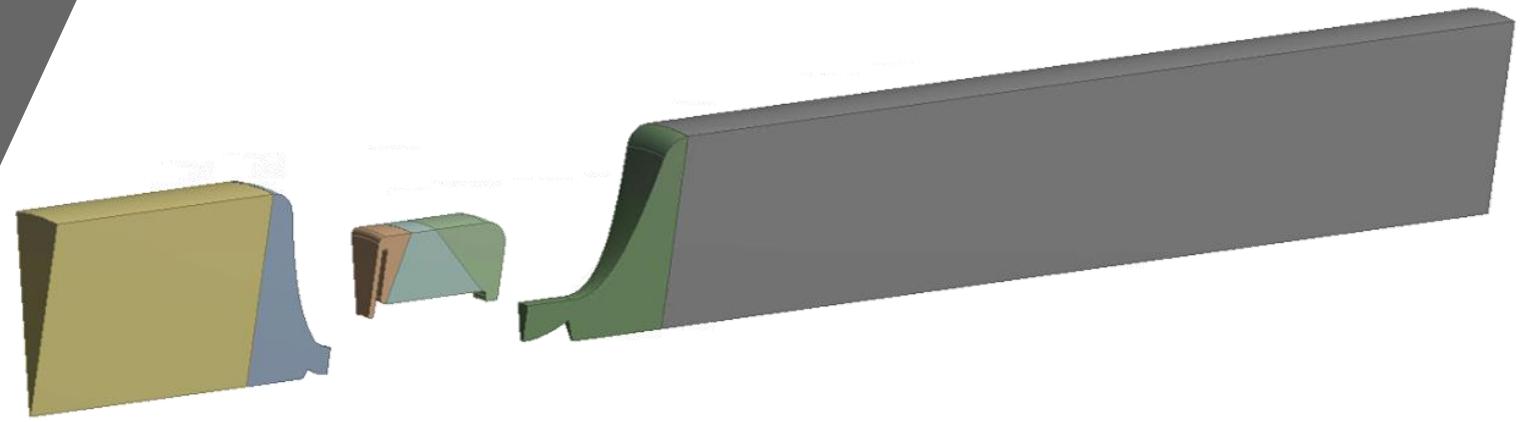
Proposed Methodology

- Unsteady Reynolds-averaged Navier-Stokes (URANS) 3D CFD simulations, a single mesh used for the entire flow domain, from intake to exhaust, of the engine.

- The only inputs required for the CFD simulations are the freestream conditions at cruising altitude and the fuel input

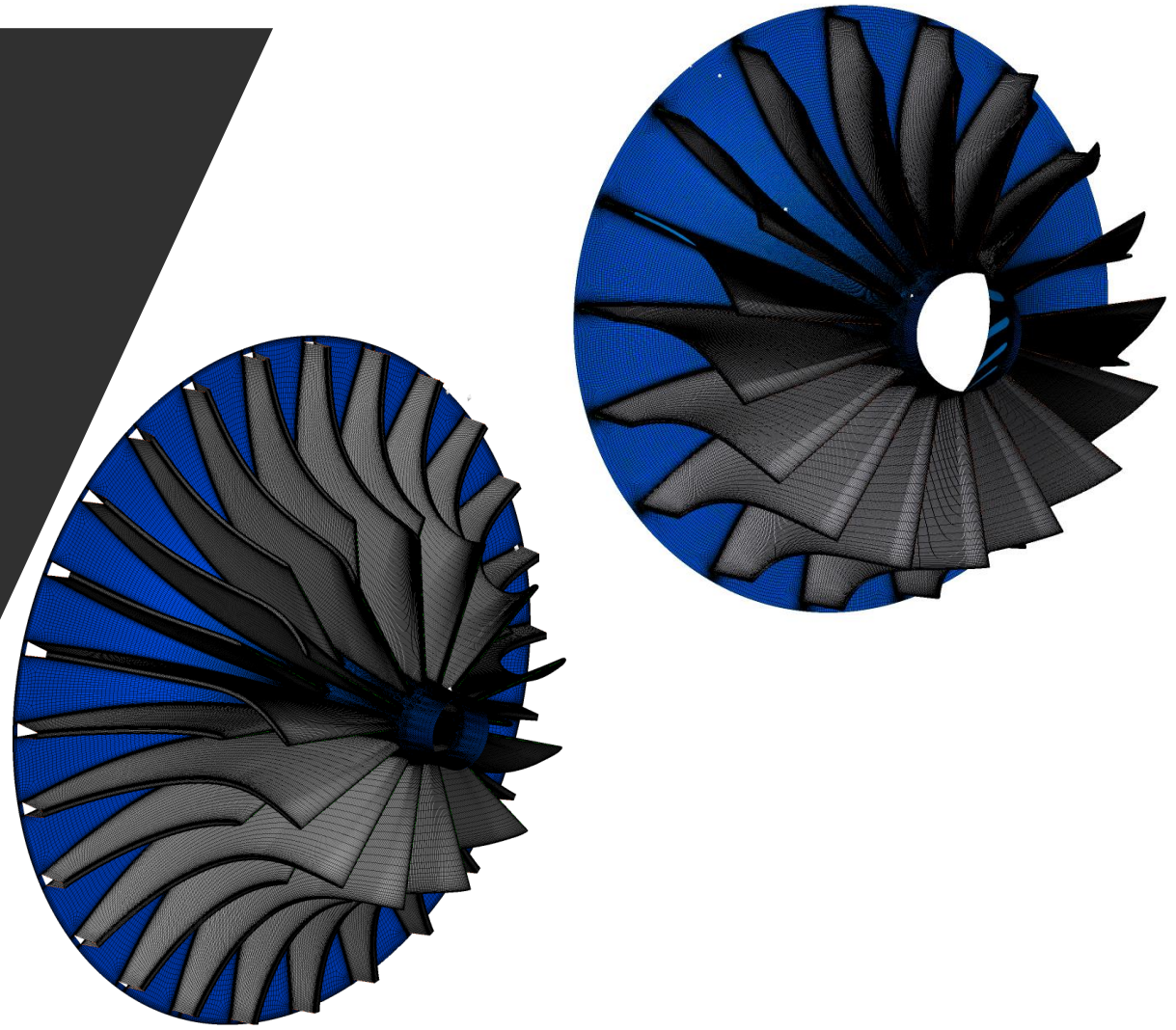
Simulation Setup

- The commercial CFD software ANSYS CFX is used
- The computational domain includes one blade passage for each of the compressor and turbine blade row
- 360º/20 sector for combustion chamber, inlet and outlet domains are also employed to reduce mesh size



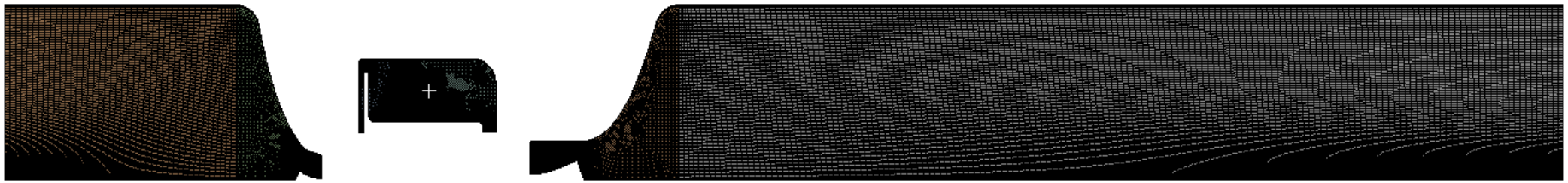
Simulation Setup

- The 3D mesh for the compressor and turbine blade rows was generated using Turbogrid, a meshing software to produce high quality mesh for blade passages in rotating machinery.

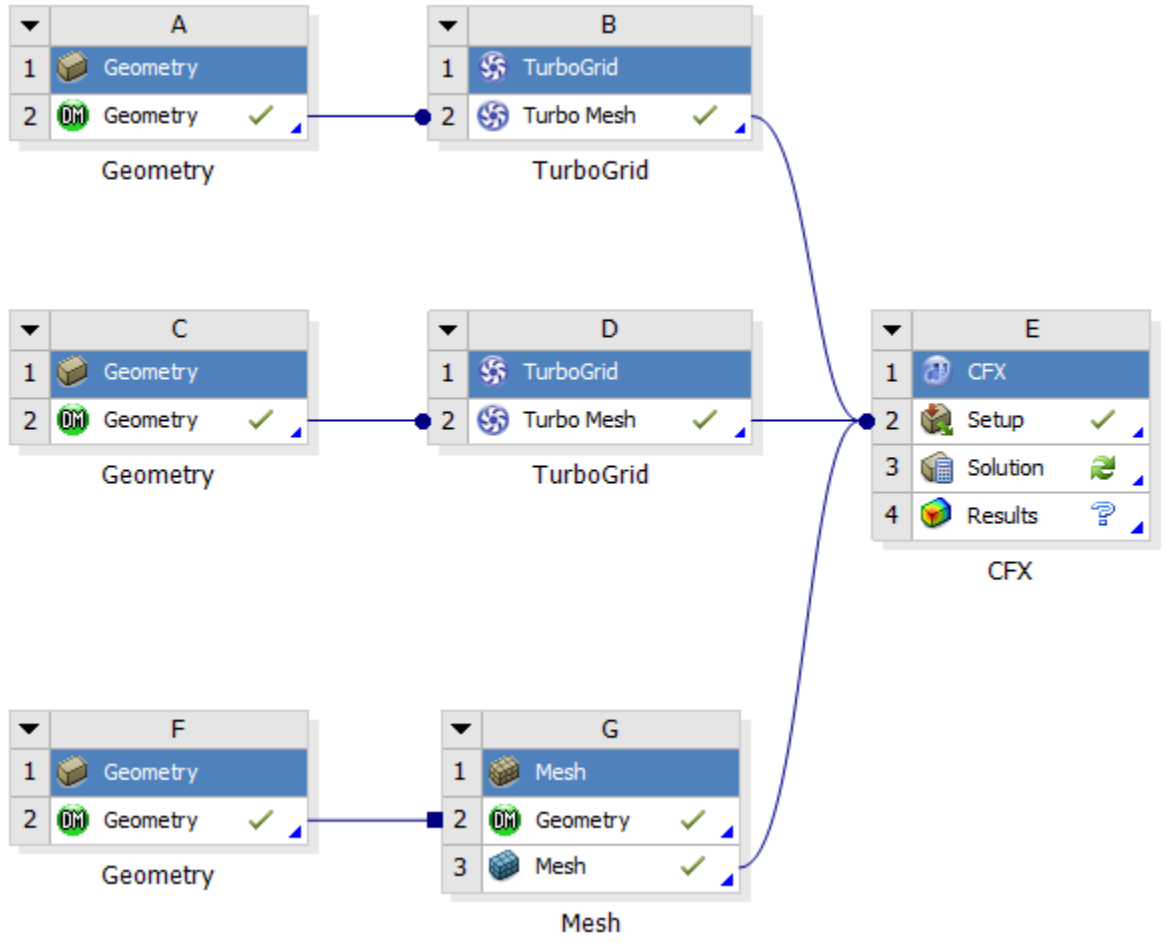


Simulation Setup

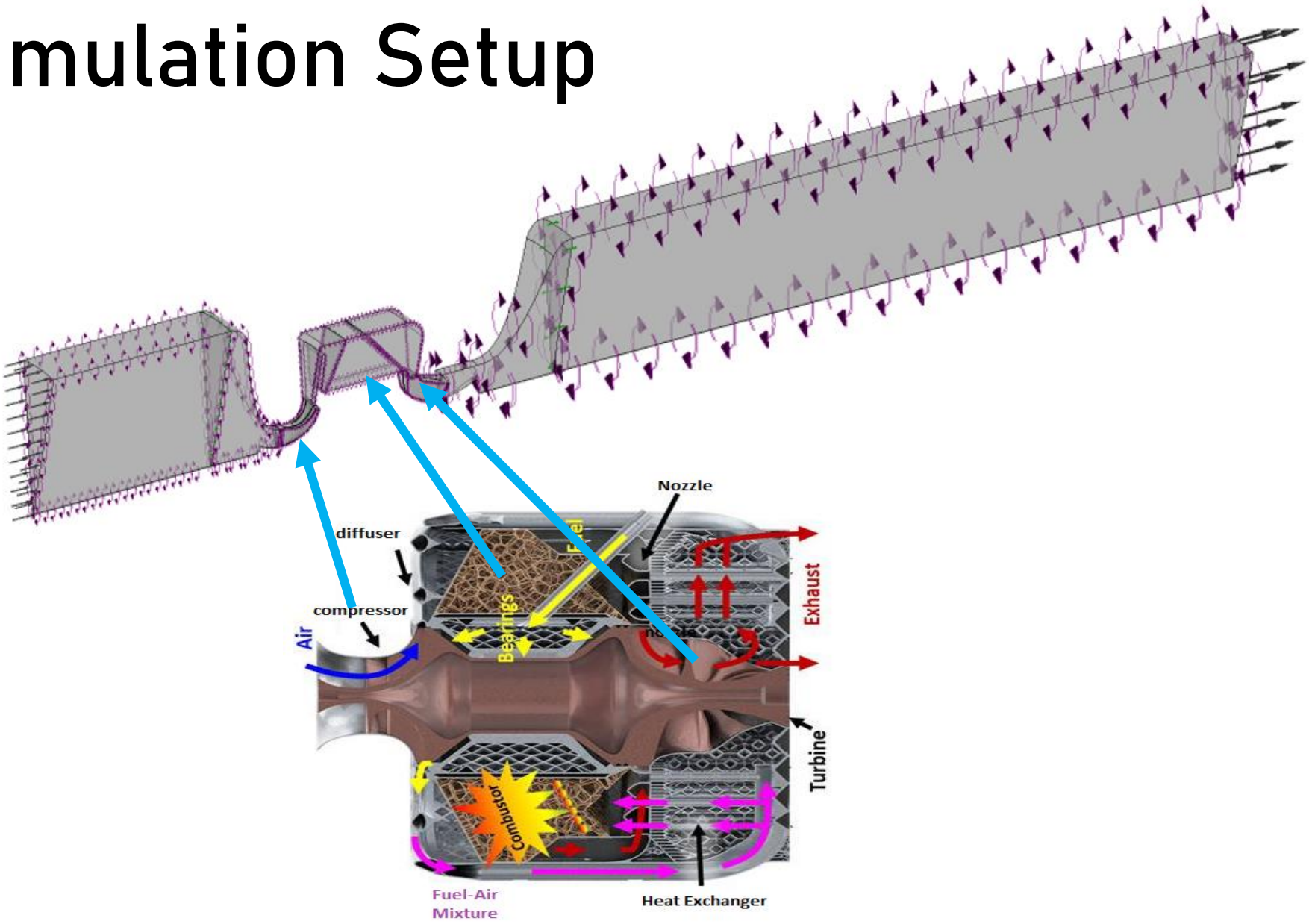
- The mesh for combustion chamber and inlet/outlet domains are generated with ANSYS Mesher:



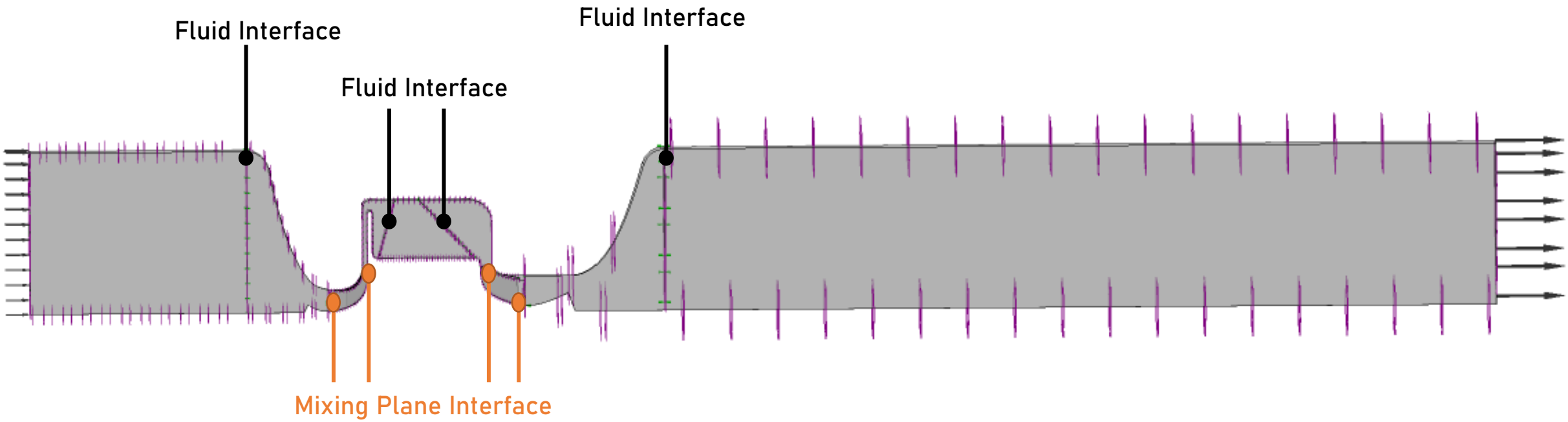
Simulation Setup



Simulation Setup

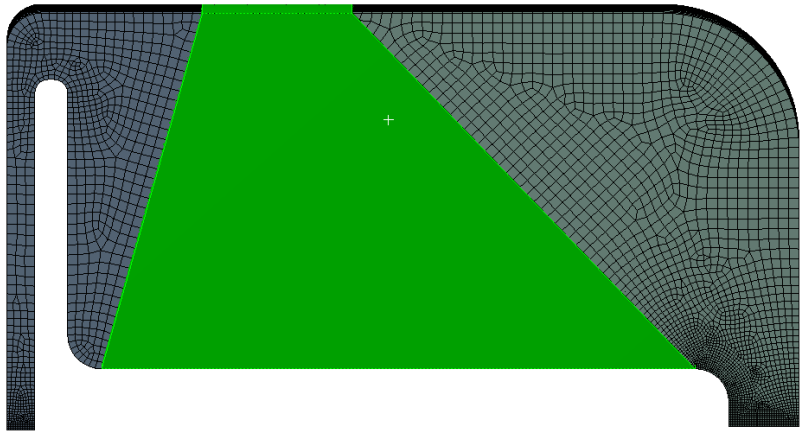


Simulation Setup



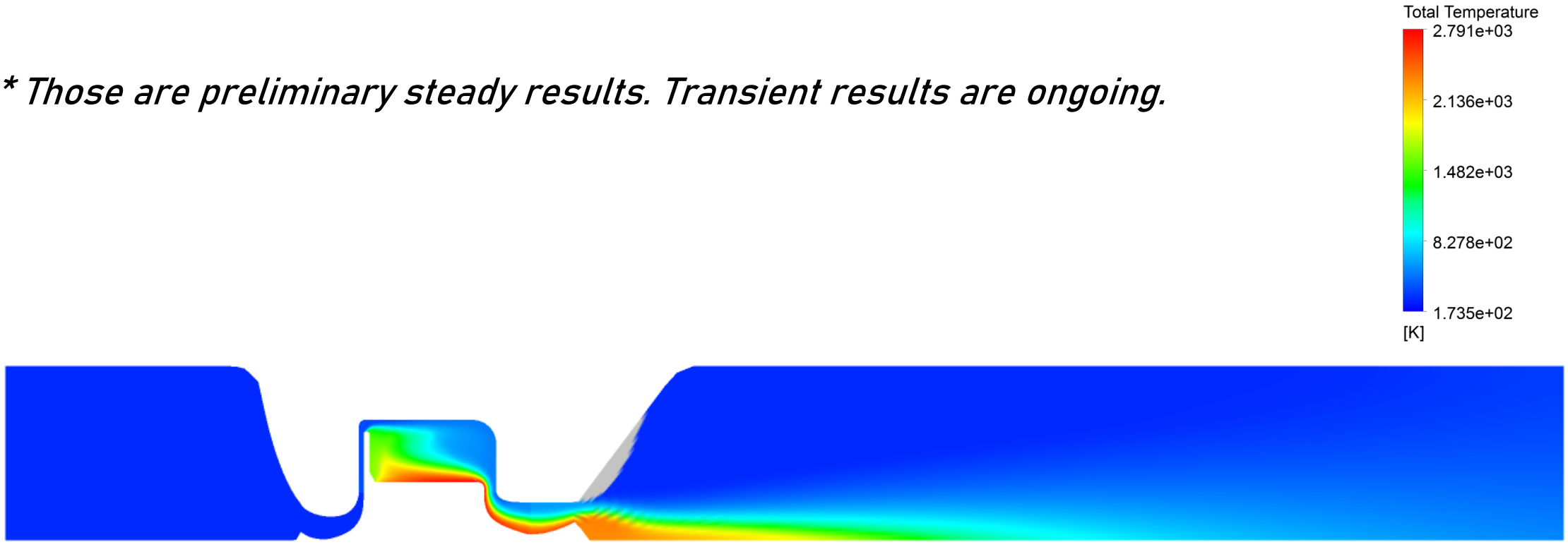
Simulation Setup

- A source for the domain corresponding to the porous media in combustion chamber was defined to obtain a temperature of 1200 K° at the turbine inlet.



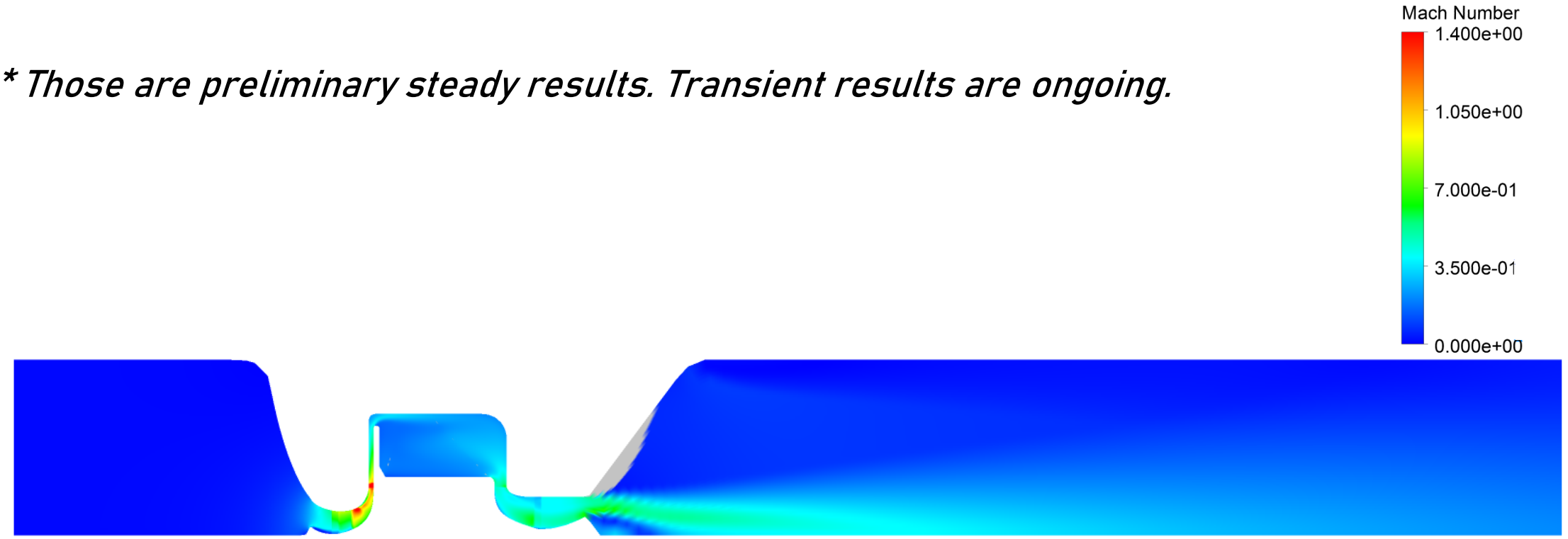
Results and Analysis

** Those are preliminary steady results. Transient results are ongoing.*



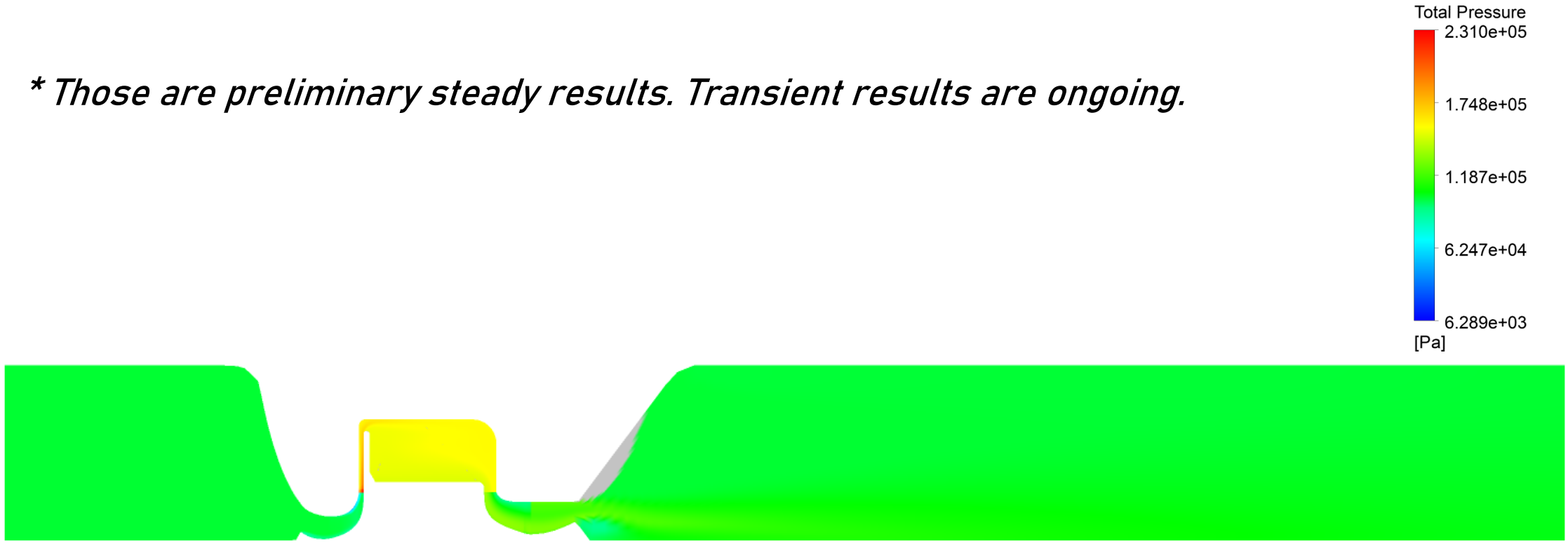
Results and Analysis

** Those are preliminary steady results. Transient results are ongoing.*



Results and Analysis

** Those are preliminary steady results. Transient results are ongoing.*



For Future Research

Porous combustor will be modeled