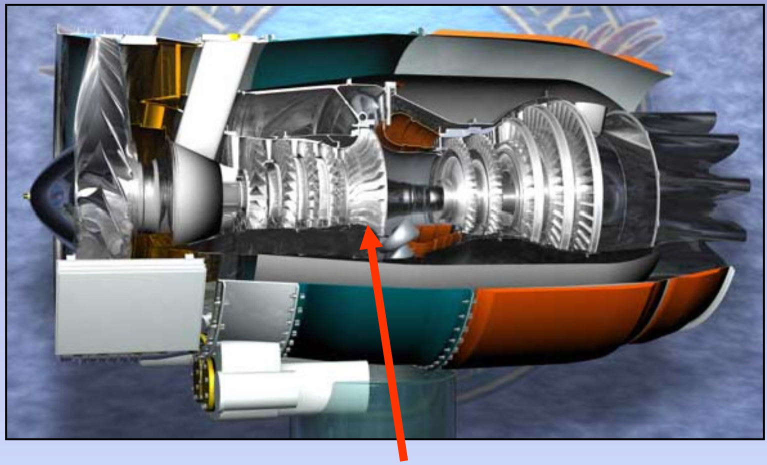
NUMERICAL INVESTIGATION OF A CENTRIFUGAL COMPRESSOR STAGE FOR AERO-ENGINES

Background

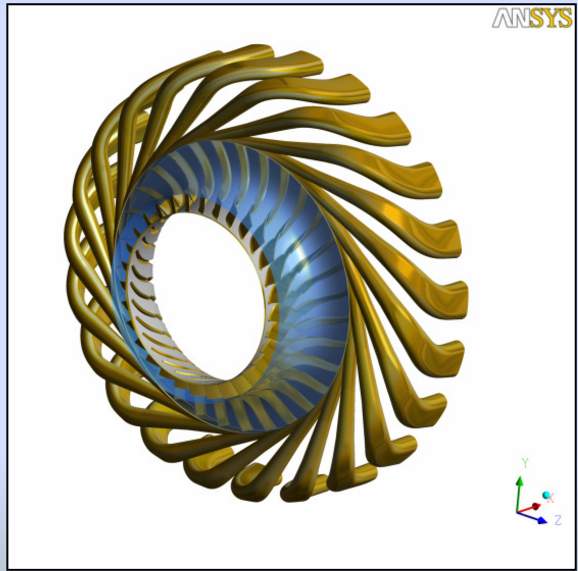
Recent design improvements in turbomachinery draw upon computational fluid dynamics (CFD) techniques. The unsteady and complex flow in these components tests the limitations of these numerical techniques.



Centrifugal Stage

Objective

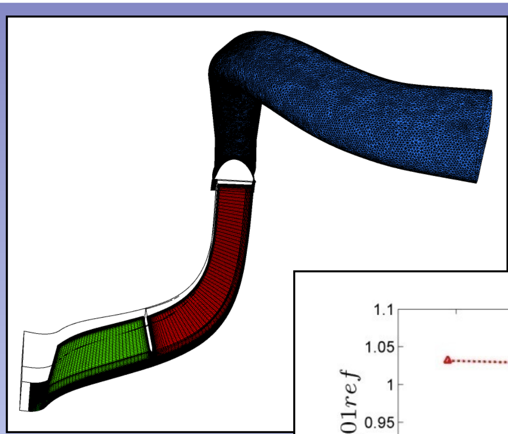
To undertake a CFD study of the flow field of a compressor stage with a tandem bladed impeller and pipe-diffuser for aero-engine applications.



Centrifugal stage consisting of a tandem impeller and pipe-diffuser

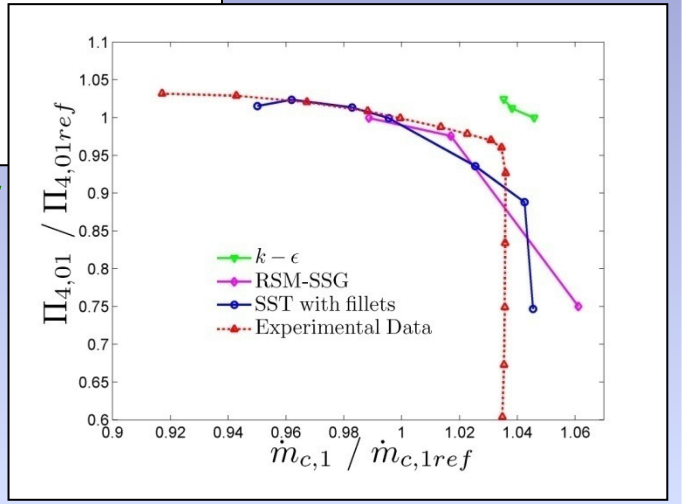
Research Being Carried Out

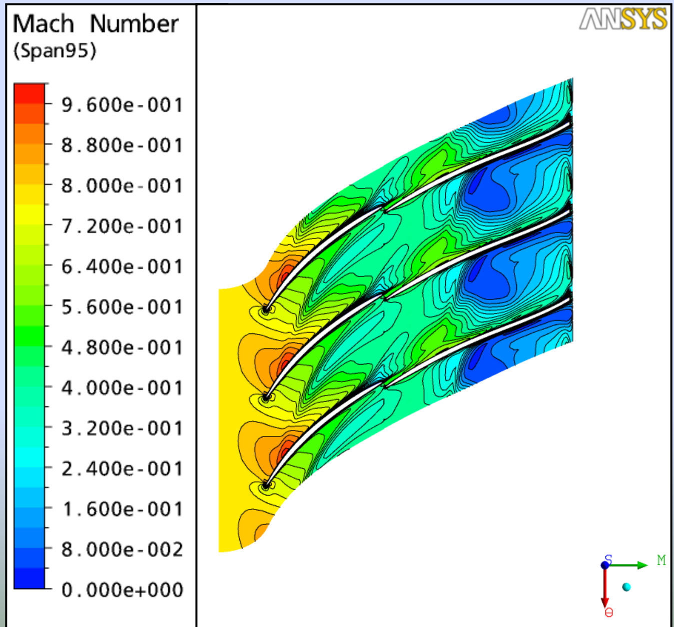
- Evaluation of appropriate turbulence modeling techniques.
- Assessment of aspects of more accurate grid geometry, such as a better fillet representation and the modeling of an exit plenum.



← *Computational grid*

Experimental and numerical compressor characteristic →





Predicted blade-to-blade Mach number contours at 95% span using SST turbulence model

Expected Outcomes

- Determination of the best design practices for CFD to predict the complex flow field in a centrifugal stage.
- Evaluation of the numerical methods used based on LDV measurements.