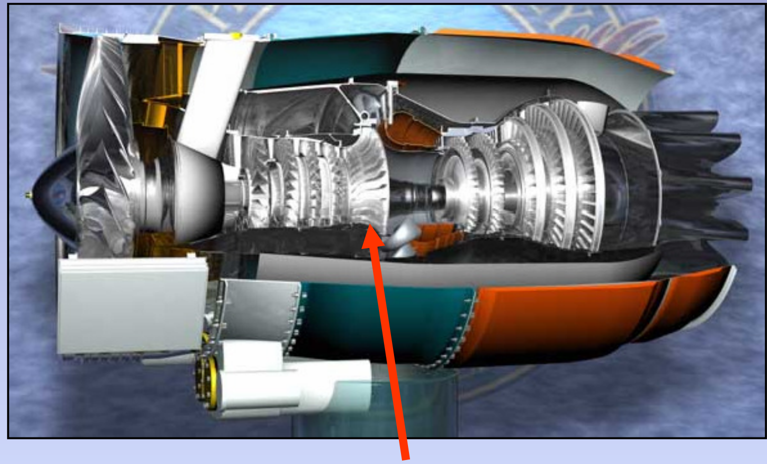
NUMERICAL INVESTIGATION OF A CENTRIFUGAL COMPRESSOR STAGE FOR AERO-ENGINES

Background

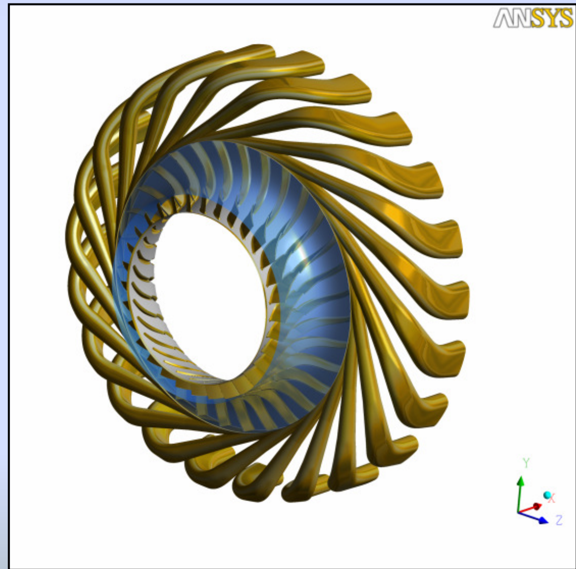
Recent design improvements in turbomachinery draw upon computational fluid dynamics (CFD) techniques. The unsteady and complex flow in these components tests the limitations of these numerical techniques.

Objective

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



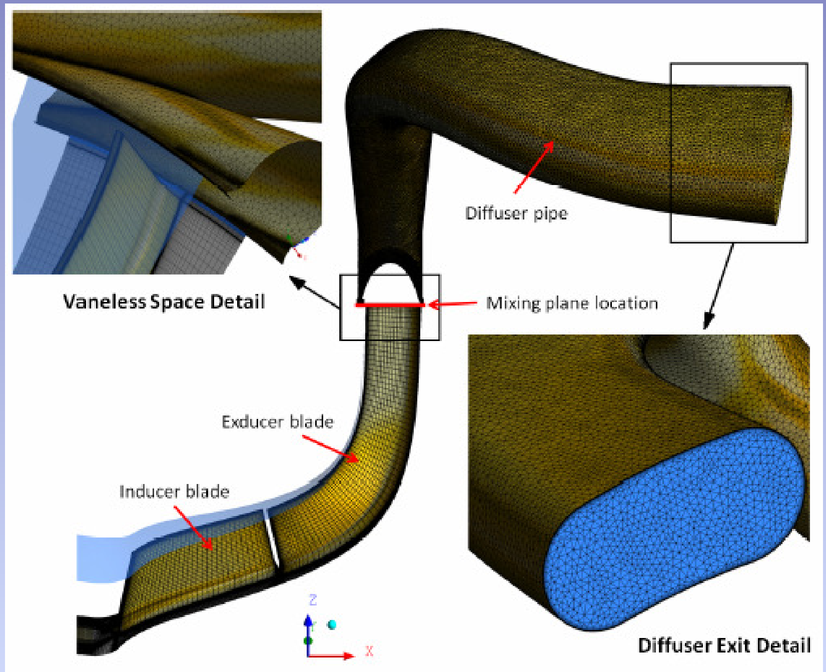
Centrifugal Stage



Centrifugal stage consisting of a tandem impeller and pipe-diffuser

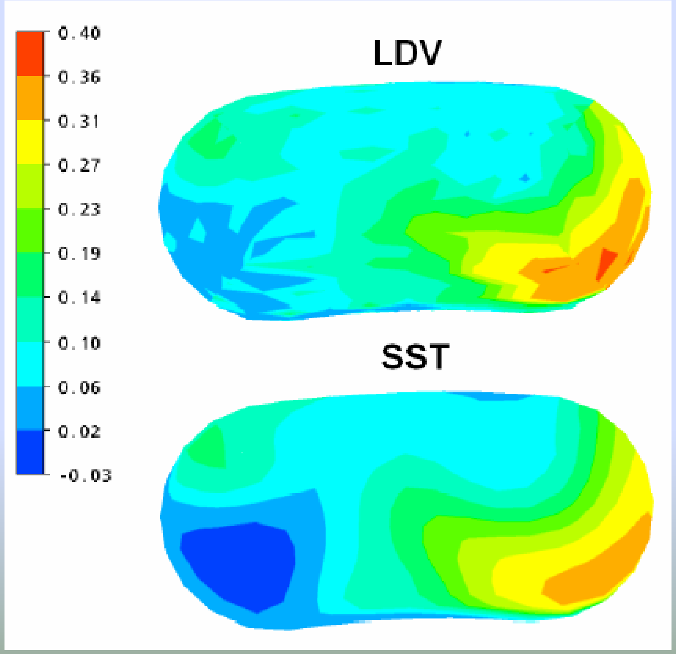
Research Carried Out

- Evaluation of different turbulence modeling techniques and comparison with experimental data.
- Assessment of aspects of more accurate grid geometry, (e.g. better fillet modeling, modeling of an exit plenum).



Computational grid

Experimental (LDV) and CFD (SST turbulence model) diffuser exit axial velocity distribution showing excellent agreement



Outcomes

- Best turbulence model and its optimal implementation for design practice identified.
- Quantitative evaluation of the CFD models by detailed LDV flow field experiments and other compressor data.