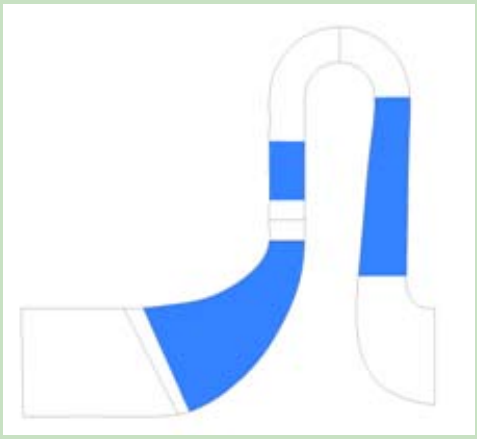
turbodesign
cfd

information

TURBOdesign CFD is a full 3D CFD solver developed for analysis of compressible and incompressible turbomachinery flows such as Compressors and Turbines or Pumps and Fans.

TURBOdesign CFD has been specifically developed to streamline the application of CFD in the blade design process and provide a faster design cycle for all turbomachinery components.



Turbomachinery Specific CFD

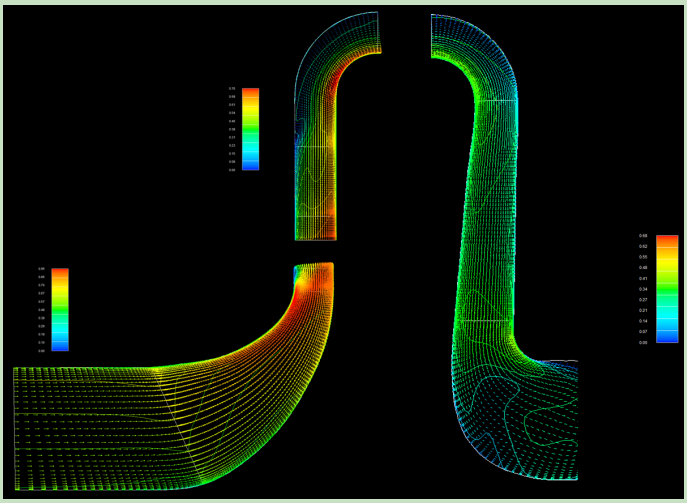
TURBOdesign CFD features turbomachinery specific pre-processing and post-processing capabilities to allow for fast setup of computational cases for single or multiple blade rows analysis at both design point or over a wide range of mass-flows (characteristics).

Application specific post-processing for each blade row and stage summary files provides at glance information on the critical performance of the machine while 3D plots allow designers to deeply investigate the flow fields in the blade channel and between stages.

Faster Product Development

TURBOdesign CFD is directly integrated within TURBOdesign Suite environment allowing seamless passage of design data from geometry generation to 3D CFD analysis.

Automatic meshing capabilities, mesh statistics and advanced refinement controls provide both novice and experienced users with a suitable platform.



Competitive Advantage

- Full integration with TURBOdesign Suite – Leading design for turbomachinery
- User friendly and intuitive interface based on a step-by-step setup process
- Solver parallelization for single blade row analysis in less than 10min on 4 CPUs
- Automatic computation of characteristic and multiple case batches
- Relevant information such as head breakdown due to cavitation in pumps or broadrband noise in fans are provided automatically from the fast RANS computations.

TURBOdesign CFD Pre-Processor

The process of generating a test case in TURBOdesign CFD consists of 5 main steps:

Step One - Input the Blade Geometry

The pre-processor has been designed to automatically import geometries from TURBOdesign Suite.

Step Two - Mesh Generation

The code supports a robust automatic mesh generation function and also allows for advanced controls for extension to stationary parts, grid flaring and tip clearance. Automatic elliptic smoothing can also be applied to provide high mesh quality on highly staggered geometries

Step Three - Specification

The specification window allows users to select turbomachinery application specific inputs and outputs along with fluid and gas properties and turbulence models

Step Four - Boundary Conditions

The Boundary Conditions window allows users to select inflow and outflow boundary conditions and turbulence parameters along with single point or characteristic definition

Step Five - Convergence & Advanced Settings

Advanced settings are available to fine tune the solver convergence parameters and parallelization

TURBOdesign CFD Post-Processor

1. **Turbomachinery specific summary files:**

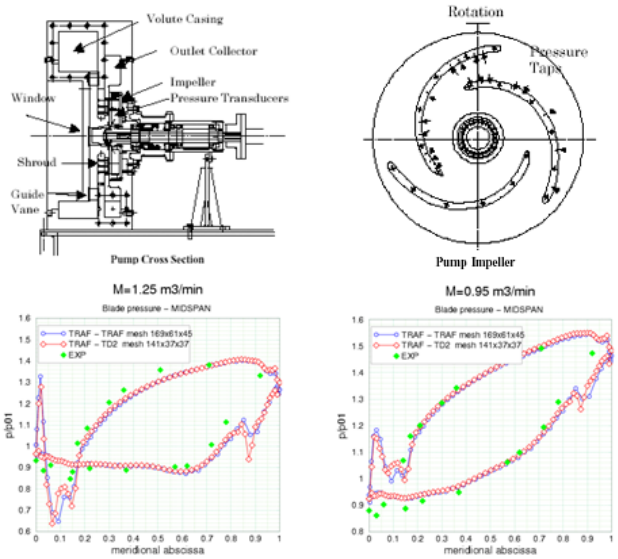
   Summary files specific to each turbomachinery application and stage components are automatically output for each case at the end of the analysis to allow for immediate post-processing of the most relevant information.

2. **Flow visualization:**

   Detailed post-processing and flow visualisation through 3D, Blade-to-Blade or Contours and Line Plots allows detailed investigation of the flow behaviour

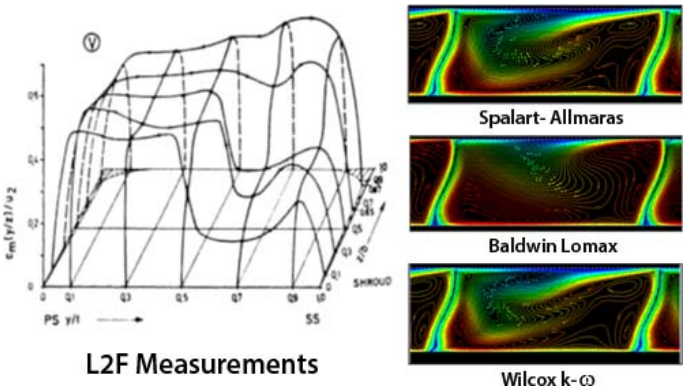
3. **Blade Loading :**

   Automatically saves the blade loading of an existing geometry in a format compatible with TURBOdesign1 input.



Solver Features and Capabilities

1. **TRAF Code:** With over 20 years of solver development and more than 80 peers reviewed papers, the TRAF code is one of the most employed turbomachinery specific CFD solvers .
2. **Robustness:** Unique features of the TRAF solver include the capability to solve the simulation down to the boundary layer which avoids usage of wall functions providing more accurate results.
3. **Computational Speed:** The code employs Local Time Stepping, Residual Smoothing and Multigrid technique is based on the Full Approximation Storage (FAS) schemes to provide unparallel calculation speed: typical computational time for a single blade row, 0.5 M Mesh is less than 10 minutes on 4 CPUs. For a similar mesh size on a single core the code is twice faster than equivalent general purpose CFD code.
4. **Turbulence Models:** TURBOdesign CFD provides a choice of turbulence models such as algebraic Baldwin-Lomax model, the one-equation Spalart-Allmaras model and the two-equation Wilcox's k-ω model, making it suitable to turbomachinery applications across a wide range of Reynolds numbers.



Advanced Design Technology Ltd
Dilke House 1 Malet Street London WC1E 7JN United Kingdom
Phone.+44 (0) 20 7299 1170
Fax +44 (0) 20 7636 8028
sales@adtechnology.co.uk
www.adtechnology.co.uk

Advanced Design Technology