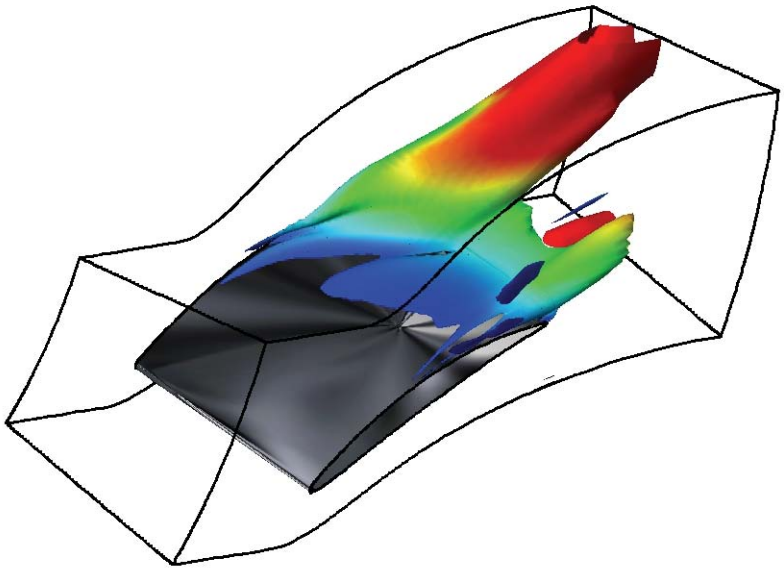
Innovative Turbulence Modeling: A Transition Model in ANSYS® CFX®

Technical Brief

World's First Turbulence Transition Model for Commercial CFD

Accurate prediction of transition between laminar and turbulent flow during 3-D CFD simulation can be accomplished using the Predictive Menter-Langtry γ–θ laminar-turbulent transition model™.

ANSYS CFX software is the first commercial computational fluid dynamics (CFD) code with reliable and calibrated 3-D transition prediction capability. Using CFD to determine the transition location between laminar and turbulent flow is critical to improving efficiency and/or longevity of equipment in turbomachinery, aerospace, marine and many other industries.



Together with General Electric Company's Global Research, ANSYS, Inc. has developed an accurate and reliable transition model for wall-bounded flows. Because this innovative approach is based strictly on local variables and standard transport equations, it delivers good performance on 3-D unstructured grids and on both single-processor machines and in parallel.

Location and extent of transition are important in most wall-bounded flows at low to medium Reynolds numbers, in which the wall shear stress or wall heat transfer are of interest. An important application is turbomachinery flows, in which blockage and loss used to assess performance are strongly affected by transition.

In aerospace applications, transition can influence the separation behavior of boundary layers and, therefore, can affect the performance of airfoils and bluff bodies. For hypersonic vehicles, transition has a significant influence on the design of the thermal protection system and allowable flight trajectories. Performance, weight and costs associated with aerospace vehicles can be optimized by accounting for transition in the design process.

The transition model in the ANSYS CFX code offers a flexible environment for engineering transition predictions that require no special provisions for geometry or grid topology.

Technical Brief

Turbulence Modeling in ANSYS CFX Software

ANSYS CFX contains a wide variety of turbulence models to provide accurate and computationally efficient results for almost every application. The widely proven SST turbulence model available in ANSYS CFX software offers significant advantages for non-equilibrium turbulent boundary layer flows and heat transfer predictions. It provides excellent answers on a wide range of flows and near-wall mesh conditions. In addition, the ANSYS CFX product contains a unique, state-of-the-art detached eddy (DES) transient turbulence model that combines the efficiency of a RANS simulation in attached boundary layer regions, and the ability to compute large eddy transient structures.

References:

Geometry provided by RWTH Aachen University.

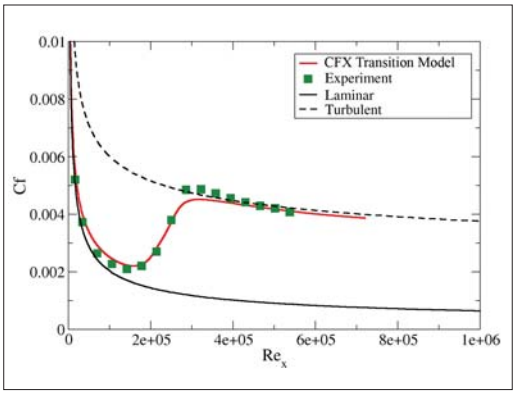
Schulz, H.D., Gallus, H.D., (1988), "Experimental Investigation of the Three-Dimensional Flow in an Annular Compressor Cascade" ASME *Journal of Turbomachinery*, Vol. 110, October.

Zierke, W.C. and Deutsch, S., (1989), "The measurement of boundary layers on a compressor blade in cascade - Vols. 1 and 2," NASA CR 185118.

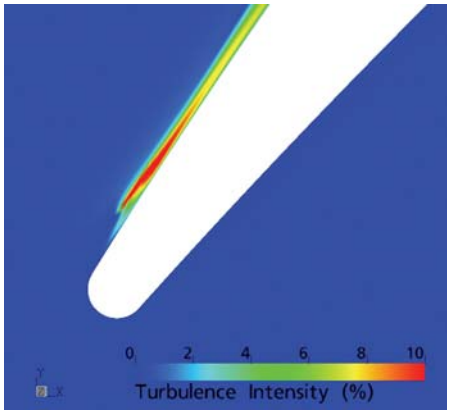
www.ansys.com

Test Cases

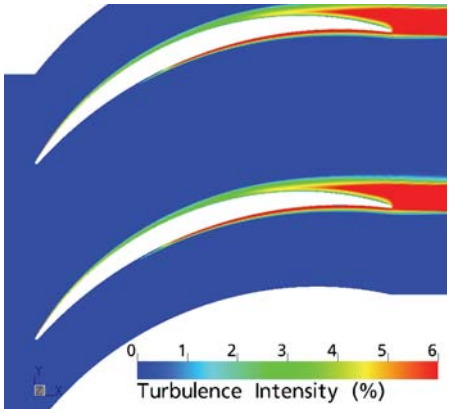
A significant number of test cases have been performed in order to test the transition model for both turbomachinery and aerodynamic applications. This includes the drag crisis of a cylinder, separation-induced transition on a circular leading edge and natural transition on a wind turbine. Turbomachinery test cases include a highly loaded compressor cascade, a low-pressure turbine blade, a transonic turbine guide vane, a three-dimensional annular compressor cascade and unsteady transition due to wake impingement. In all cases, extraordinarily good quantitative agreement with experiment was achieved.



Comparison between experimentally measured skin friction coefficient and the computed value with the ANSYS CFX transition model for a flat-plate boundary layer undergoing bypass transition



Computed turbulence intensity around a highly-loaded compressor blade.



Leading edge separation-induced transition.

Summary

The transition model in ANSYS CFX software has been tested extensively and has been proven essential in determining the location and extent of transition in both aerospace and turbomachinery applications. The ANSYS CFX product contains the world's first commercial model to predict transition between laminar and turbulent flow during 3-D CFD simulation.

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
USA
724.746.3304
ansysinfo@ansys.com

Toll Free USA/Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com

cfx-transition_ Dec06

GSA Contract Holder