

Helicopter Rotor-Airframe Interaction

In this example, a helicopter in a forward flight is modeled using FLUENT. The spinning rotor blades are simulated using a lumped parameter model. This model uses momentum sources that are derived from the blade geometry and local flow conditions. When used to predict the flow field through the rotor plane, the model shows good agreement with experimental data.

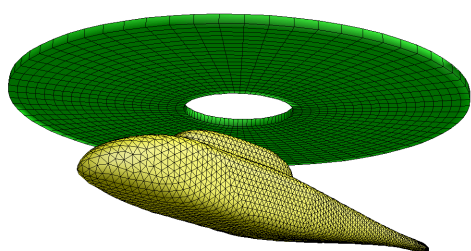


Figure 1: The surface grid used for the simulation

Analysis of the complex flow field around the rotor and fuselage of a helicopter provides a firm base for efforts for improving upon the design of these components. In the present study, FLUENT was used to analyze the flow around a model helicopter design using a novel treatment for the rotor system. The complex aerodynamic interaction between the spinning rotor and stationary airframe was solved by coupling blade element theory with the governing equations of the flow. The model was tested successfully against experimental data for two cases, one of which is reported upon here: the 2 Meter Rotor Test System, or 2MRTS, which is tested in the 14' x 22' subsonic wind tunnel at the NASA-Langley Research Center.

The 2MRTS is a general purpose rotorcraft model testing system. It is derived from the Rotor-Body Interaction, or ROBIN configuration. This configuration consists of a fuselage and a four-bladed fully articulated rotor system, which, under normal conditions, operates with a rotor speed of about 2000 rpm. In the CFD simulation, the rotor is modeled as a set of momentum sources applied to a cylindrical disk that corresponds to the swept out volume of the spinning blades. To obtain these source terms, lift and drag forces must be computed for each discretized blade segment as functions of the angle of attack, geometry, and blade speed.

The surface grid on the fuselage and on the cylindrical disk representing the spinning rotor are shown in Figure 1. The surface mesh on the fuselage is created using triangular elements. The rotor disc zone is discretized using hexahedral cells. Tetrahedral cells are used for the remainder of the volume mesh, with pyramids used in the transition region.

The pressure field on the fuselage

and in the vicinity of the rotor are shown in Figure 2. The high pressure at the leading edge of the aircraft suggests forward motion. The pressure fields above and below the rotor are the result of the applied momentum sources, and illustrate the lift generated by the rotor model.

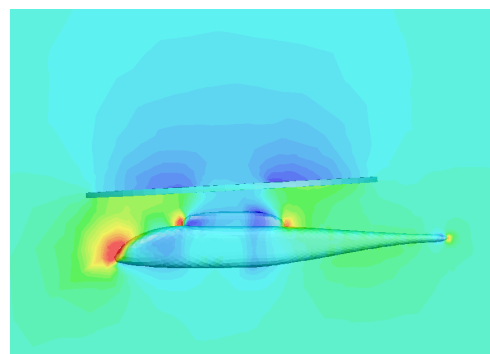


Figure 2: Pressure contours on the aircraft and surroundings

The induced inflow ratio normal to the tip path plane, λ , is a computed quantity that allows for a comparison between the simulation results and experimental data. This dimensionless value is the difference between the normal component of air velocity passing through the tip of the rotor blade and the normal component of air

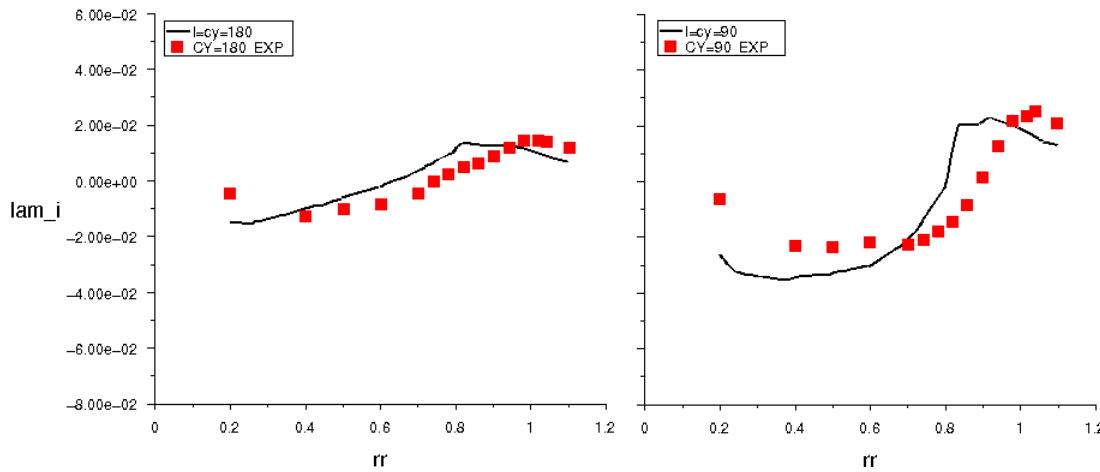


Figure 3: The induced inflow ratio normal to the tip path plane as a function of normalized radius along lines that extend from the rotor center to the front of the craft (left) and to the side of the craft (right)

velocity in the free stream, normalized by the tip speed. Experimental measurements of these velocities are accomplished in the wind tunnel using a two-component laser velocimeter. In Figure 3a, FLUENT results for lambda (black) as a function of normalized radius are compared to experimental measurements at the 180 degree line, which extends from the rotor to the front of the craft. (See Figure 4.) In Figure 3b, the comparison is along a trace at 90 degrees, which extends from the rotor center to the side of the rotor plane. The trend in the data is well matched

by the FLUENT results for both cases.

In Figure 4, the ratio of induced normal inflow for the entire rotor disk area is shown. The results shown in Figures 3a and 3b above are indicated by patterns in the contour plot that are along traces that extend from the rotor center to the left, and from the rotor center towards the top in this figure.

The rotor model has been validated successfully in another rotor-airframe simulation as well. In that case, a comparison of

surface pressure on the fuselage was validated along with the velocities in the vicinity of the rotor disk. In Figure 5, the rotor model is used for the simulation of a more realistic helicopter configuration. In the figure, pathlines representing the induced airflow in the rotor region are shown along with surface pressure on the fuselage.

In summary, a novel approach to modeling a helicopter rotor has been developed and validated in FLUENT. This model uses momentum sources to simulate the complex aerodynamics of the spinning rotor. Numerous comparisons with data, some of which are reported here, have been very successful. The model can be easily applied to other rotor configurations, and serves to greatly simplify the task of in-flight helicopter simulation with CFD.

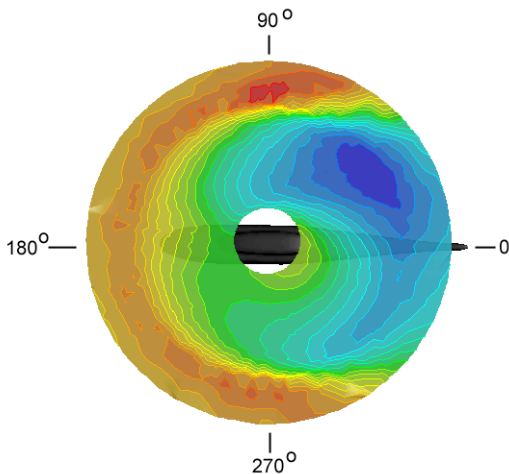


Figure 4 : Contours of the ratio of induced normal inflow on the entire cylindrical disk representing the rotor plane

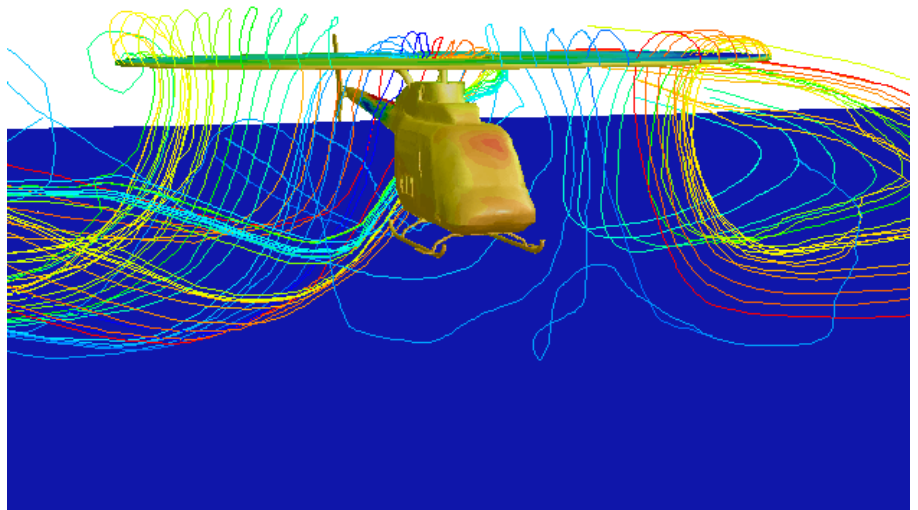


Figure 5 : The rotor model has been successfully applied to other helicopter simulations, such as this realistic model, where pathlines show the flow through the rotor