Report of EUFAR Expert Group on Airflow Disturbance

CIRA, Capua, Italy.
16-20 September 2000

by

Philippe Nacass
Working Group Coordinator
Météo France, Centre d'Aviation Météorologique,
Brétigny-sur-Orge (France)

with

Markus Hermann and Frank Stratmann
Institut für Troposphärenforschung,
Leipzig (Germany)

and

Cindy Thowy
National Center for Atmospheric Research, Boulder (Colorado, USA)
and Oregon State University, Corvallis (Oregon, USA)

INTRODUCTION

On 18 and 19 September 2002, the first EUFAR Workshop on "**Airflow Disturbance**" was held at Capua (Italy) .This meeting was jointly organized with three others EUFAR Workshops : "Microphysics Liquid Phase", "Microphysics Ice Phase" and "Small Scale Turbulence" hosted for a week (16-20 September) by the **Centro Italiano Ricerche Aerospaziali** (CIRA).

The four workshop schedules were merged ; so the participants were able to listen every communications in every workshops. More than thirty scientists from twelve countries in Europe (France, Germany, Italy, Poland, Spain, United Kingdom) and overseas (Australia, Canada, Israel, Mexico, Russia and USA) attended these meetings.

SOFTWARE COMPARISONS

Five months ago, during the EUFAR Workshop on "Airborne Particle Inlets" held at Leipzig (April 2002), it had been discussed the necessity to have comparison exercises between models of airflow disturbance using home made or commercial softwares. A common test was proposed before the "Airflow Disturbance" Workshop.

Even if this request had to help the Inlet-users community and gave a good base for discussion at Capua, it cannot be realized due to the short delay and the amount of work for pre-processing, processing and post-processing. But this test may always be computed by the attendees for a later comparison.

COMMUNICATIONS

This Workshop began on 18 September with two general communications showing examples of the Computational Fluid Dynamics (CFD) evolution for the last twelve years.

- Cindy Twohy (NCAR/OSU, USA) gave a review of past and present airflow studies at NCAR. Potential flow were used to estimate particle behavior around aircraft fuselages (King-Air, Electra and C-130). Now, modern CFD packages (Star-CD and Fluent) are used for simulation around aircraft parts as wing, pod, sampling location (C-130 and Gulfstream-V) and several instruments (counter flow virtual impactor, cloud water collector, …).

- Philippe Nacass (MF, France) showed also simulations computed by the same potential flow code around instruments (anemoclinometer, radome and boom) and aircraft (Merlin-IV and Fokker-27). Then, simulations with a CFD package (Gambit 2.0 and Fluent 6.0) are presented : around and in airborne instruments (sonic anemometer, Gerber probe, isokinetic intakes, FSSP, Radar,…), around partial or complete aircraft fuselage (Merlin-IV, Casa 235) and even around a full research vessel. Path lines and trajectories are simulated and animated by seeds or filaments.

The second day, four specialized communications were presented ; the most of them were focused on modeling of particle inlets and were not to far of those presented at the EUFAR International Workshop on Airborne Particle Inlets at Leipzig (12-13 April 2002).

- Frank Stratmann (IT, Germany) described a home-made fine particle model programmed to be a User Define Function (UDF) for the Fluent commercial code. This is an Eulerian particle dynamics model, with simulation of particle formation, transport, transformation and deposition processes. Several applications are shown.

- Markus Hermann (IT, Germany) used Gambit 2.0 and Fluent 6.0 for comparison between experimental measurements and 2D simulation calculation of an aerosol inlet flying on a Gulfstream I. Turbulence equations in the model are "k-ε RNG" (ReNormalization Group theory).

- Markus Hermann (IT, Germany) used the same softwares for modeling an aerosol inlet for a commercial aircraft (Airbus A340-600). The project is named CARIBIC (Civil Aircraft for Regular Investigation of the atmosphere Based on an Instrument Container). The 3D simulation run with the turbulence equations k-ε RNG.

- Philippe Nacass (MF, France) presented a part of the thesis conducted by a student at the Université de Clermont-Ferrand, Laboratoire de Météorologie Physique (LaMP). This work uses simulations to develop an airborne CVI. Simulations give the particle trajectories around the CVI tip and in the probe. The porous material is simulated. The cut diameter is calculated and particle trajectories are well represented for particle diameters smaller, near and larger than the cut diameter.

RECOMMENDATIONS AND CONCLUSIONS

On Friday 20 September in the morning, a discussion with attendees of the workshop gave some conclusions that are resumed here :

- CFD calculations are useful in predicting general characteristics of airflow and particle trajectories around aircraft and inlets.

- Details of results may vary with grid density, solution method, wall effects, turbulence parameters, boundary conditions, etc, so quantitative details of flow and collection efficiency should be verified by experiments whenever possible.

- Recent CFD & computer advances allow solution of more complex problems (e.g., whole ships or aircraft, given sufficient time!).

- From our CFD community point of view, there is a need to model the flow field around aircraft (keywords: shadow and enhancement zones, aircraft boundary layer), the flow field and particle trajectories for any kind of inlets (keywords: aspiration and transmission efficiencies) and the flow field for "free stream" measurement instruments (like e.g. the FSSP) separately for each aircraft and mounting position.

- For flow around aircraft, small particles follow streamlines, intermediate-sized (approx. 100 micron) particles partially deviate from streamlines, creating "shadow zone", with enhancement region just above and large (approx. 1000 microns) particles follow straight trajectories relatively unaffected by airflow changes.

- Issues of inlet sampling on aircraft are still an ongoing problem, especially at very high speeds (> Ma 0.7) where shocks may form around aircraft and inlets. More research is needed here, including on what happens to aerosol particles/droplets in a shock region. Side comment not discussed at workshop : inlets for large (>3 micron) aerosol particles and issues of drop/ice crystal breakup on surfaces still need research as well. Breakup issue affect cloud physicists too.

- Some laboratories as IFT are making great improvements to standard commercial codes capability (Fluent). The IFT Fine Particle Model (FPM), which should be encouraged/supported and made available to the community (international if possible!).

- The other EUFAR working groups should make a statement if, from their point of view, there is a need for CFD modeling, and if yes, how this need looks like.

- The atmospheric CFD community could use help from the aerodynamics (or turbulence?) community on what turbulence parameters (i.e., intensity, length scale) make sense for different simulations, depending on model scale, turbulence models, etc

- On long-term, a center of excellence or competence (however named by the EU) for CFD modeling should be founded or named.

- There should be funding for CFD code development (e.g. with respect to particles in CFD codes), CFD result validation (e.g. comparison between models and experiments).