24 th EUROPEAN ROTORCRAFT FORUM
Marseille, France - 15th-17th September 1998

AE 01

The real Need for Navier-Stokes Computations in Helicopter Industry and Requirements for an Efficient Analysis

Nannoni, F., and Righi, M.,
Aerodynamics and Flight Mechanics Department
AGUSTA Un'Azienda Finmeccanica

Abstract

The issue of exploitation of Navier-Stokes solutions in helicopter design is discussed. Most internal flows are currently successfully and effectively investigated by means of Navier-Stokes equations while helicopter flowfields concerning fuselage and rotors are still quite a cumbersome problem to solve by CFD. Helicopter applications range from fuselage drag predictions from qualitative analysis of tilt rotor induced secondary flows. Basically, industrial applications of academic or research codes give accurate but not yet fully reliable results at very high expenses of training, mesh preparation and CPU time, while engineering applications use Navier-Stokes solution to shed some light on unknown flow patterns. Both approaches are evaluated very positively, showing that interest in Navier-Stokes equations is well justified but also that this kind of CFD analysis is a very sophisticated and tricky design tool which, to be effective for industry, must be used very carefully.

1 Introduction

Numerical techniques in aerodynamics have become more and more effective together with computer power growth. As a consequence, design tools are showing an increasing level of complexity from the point of view of both physical and mathematical models. In the aerospace industry, as far as aerodynamics is concerned, simplified "engineering" methodologies, based on empyrical data correlations, can now be compared to panel methods, *full potential*, Euler and Navier-Stokes solution based methodologies.

It is well known that helicopter design has to put up with problems which appear to be far more cumbersome than fixed wing design. A great deal of aerodynamic design work is done with the aid of wind tunnel experiments, leaving to numerical methods, most of which still based on simplified formulations, the task of preliminary analysis and wind tunnel testing campaign scheduling. All this usually gives a very expensive and lenghty design phase.

Navier-Stokes solution are very popular in helicopter industry, as witnessed by joint research programmes (HELIFUSE [5]) and several scientific papers. Apart from image purposes, the issue of actual exploitation of Navier-Stokes results in helicopter design work is treated together with an attempt of defining what kind of improvements would be best welcomed by industry.

2 The numerical solution

Navier-Stokes equations are the mathematical transcription of physical conservation laws concerning fluid motion. Equations are therefore to be considered as an *exact* model, but, as a matter of fact, approximations are introduced in the practical numerical solution. They concern:

1. Turbulence modelling: it does not exist a model which can efficiently account for turbulence effects in many different flows.
2. Numerical solution: discretization in both space and time causes numerical viscosity and thus artificial dissipation and diffusion.
3. Boundary conditions: specification of pressure and temperature at a solid wall can only be approximated.
4. Initial conditions.

As a consequence of items 1 and 2 unsteady phenomena can be "averaged" in non-physical patterns. As a consequence of items 2 and 3 the computational mesh resolution and quality is of fundamental importance for results accuracy and reliability.

Navier-Stokes analysis is affected by all these uncertainties which have not negligible effects on final results: a badly calibrated turbulence model, an unproper level of upwind in the numerical scheme or, simply, a bad mesh, for instance, can be blamed for detecting an inexisting separation or for giving a drag force several times bigger than expected.
Since Navier-Stokes analysis, in comparison to simplified methodologies, are supposed to give a higher accuracy, it is easily understood that extreme care has to be taken before launching every *run*.
Note that this is true for any of the existing approaches like finite volume, finite elements or finite difference.

3 Applications

Many applications of Navier-Stokes analysis have been tested on helicopter-related flows of which examples can be:

1. isolated rotor flowfield simulation in order to give performance predictions (*e.g.* Figure of Merit)
2. isolated rotor flowfield simulation in order to give insights to new geometries effects (*e.g.* advanced blade tip shapes vs straight tip)
3. interactional effects due to rotor-fuselage coupling
4. isolated fuselage flowfield for performance purposes (evaluation of drag)
5. isolated fuselage flowfield for qualitative analysis of flow patterns

3.1 Isolated rotor

Various applications of full Navier-Stokes analysis on an isolated rotor exist (see [18], [17]). Good results, from the qualitative point of view only, have been achieved at a very high computing cost.

At the time being, however, it is not worth to use this solution for performance predictions purposes: existing performance codes give more accurate results at only a small fraction of the cost. The main problem concerns tip vortex resolution which is strongly affected by numerical diffusion. Grid refining (see [6]) helps but increases furtherly the computation costs.

3.2 Rotor-fuselage interaction

Interaction between rotors and fuselage is a very complex phenomenon in which Navier-Stokes equations can help shedding some light over.

A rotor can be represented in different ways. It can be either modeled by a real rotating grid using, for instance, a *chimera* approach or just approximated with an actuator disk-like model in which *ad hoc* momentum source terms are added to the basic equations.

$$S_x = \rho {V_{rot}}^2 c C_x \tag{1}$$

$$S_y = \rho {V_{rot}}^2 c C_y \tag{2}$$

$$S_z = \rho {V_{rot}}^2 c C_z \tag{3}$$

This approach gives a reasonable model which can easily be made time-accurate. Blade flow must

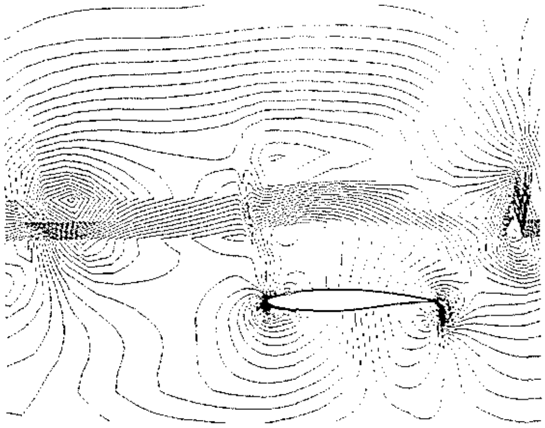

Figure 1: Isomach lines on fin-like flowfield. The momentum source terms, representing the rotor are put on a straight horizontal line.

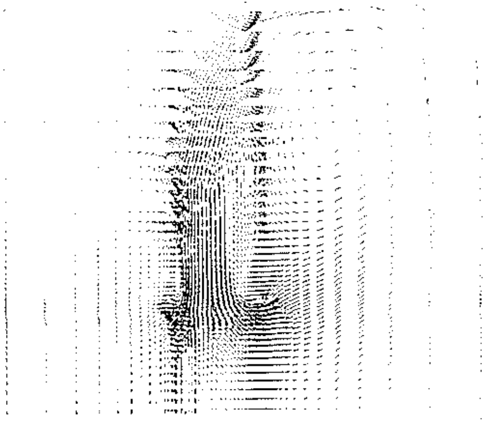

Figure 2: Velocity vectors of a rotor flowfield. The rotor is represented by momentum source terms in the lower part of the figure. The tip vortices are noticeable.

be of course approximated by table look-ups but interaction can be fully resolved. Examples of this method can be found in [3], [4], [8], [10], [13], [14], [15] and [21]. A similar approach is also followed in propeller-driven aircraft flowfield prediction, see for instance [1]. Meaningful indications on flow patterns and even acceptable performance predictions ([15]) have been achieved with fairly small models and thus acceptable computing costs (even for industry).

Some examples of calculations are reported. In figure 1 an example of rotor induced flowfield is shown. It has been obtained adding the momentum source terms (equations 1, 2 and 3) to the Navier-Stokes equations. In figure 2 a small example of computation is shown in which a "rotor" induced flowfield is introduced in the simulation of the flow around a fixed structure. All calculations (including figure 3) have been performed with a inhouse built software code based on the unsteday time-accurate solution of the compressible Navier-Stokes equations adopting finite-volume space discretisation. In both cases analysis is time accurate and shows unsteady flow patterns. The only aim of these pictures is to show how straightforward the inclusion of a rotor flowfield (although it is an approximated one) in a calculation can be.

3.3 Isolated fuselage

HELIFUSE project [5] and other scientific papers ([11], [12], [7] and [2]) have showed the capability to compute drag, through the integration of Navier-Stokes equations, on complex fuselage geometries.

As pointed out by Costes in [5], results are very encouraging, showing very good flow patterns and an impressive agreement on pressure distribution between the different solvers and the wind tunnel measurements. As far as drag force is concerned, however, the scatter becomes much wider. Considering this inaccuracy in drag force and the effort required for grid generation, the adoption of Navier-Stokes solvers in fuselage design phase may seem unrealistic at the moment.

3.4 Others

Beside this "global" applications, there are many "local" Navier-Stokes solutions oriented to give designers partial views of interesting flow patterns and quantitative predictions:
(i) internal flows like air intakes, exhausts, engine cooling or ventilation
(ii) external flows like dynamic stall analysis.

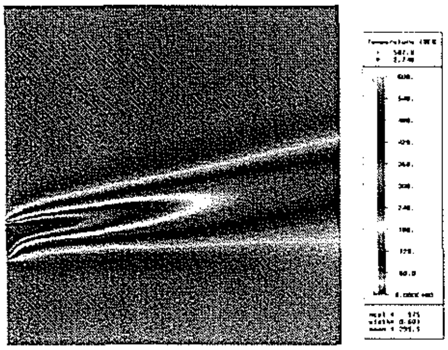

Figure 3: Temperature contours of a jet diffusing in a free stream

In figure 3 an example of a "local" application of Navier-Stokes is reported. The simulation concerns a jet diffusing in a free stream with different density and temperature. The objective was the analysis of the trajectory of exhaust gas.

While these are becoming more and more "standard" design approach, performances and aerodynamic analysis of the complete helicopter are usually left to simpler methods and wind tunnel. The reason is of course that internal flow analysis is more easily analysed by Navier-Stokes solution because of:

1. lower Reynolds number
2. less important compressible effects
3. simpler meshes which can be safely generated by automatic codes (finite elements)

As far as dynamic stall is concerned, very interesting research work is being done which will come up eventually with accurate airfoil codes and indications for dynamic stall models.

	Navier-Stokes	Rotor code
mesh	40-80 h	0
CPU time	20-40 h	10-20 sec
training	some weeks	some days
reliability hover	not yet	for conv. geom. only
reliability full flight	none	for conv. geom. only

Table 1: Comparison between Navier-Stokes and traditional rotor codes, based on lifting-line theory and experimental airfoil table look-ups on prediction of a rotor performance (total power and generated forces). Indication of CPU time is referred to a high capacity Work Station.

4 Navier-Stokes costs

Navier-Stokes analysis is very expensive. The total cost breakdown (excluding software and necessary hardware purchase) is given by: *(i)* training for mesh generator and solver, *(ii)* man power needed for mesh generation, *(iii)* CPU time needed for solution, including sensitivity analysis to mesh characteristics and numerical scheme parameters variation, *(iv)* output results analysis.

As compared to other analysis methods, most of this items are outrageously high. A panel method, for example, needs a time for mesh generation at least one order of magnitude lower, while the requested CPU time may even be over two orders of magnitude less.
As far as rotor performance are concerned, the alternative would be dedicated codes which use a simple lifting line theory and give reliable results in some seconds without the need of any computational mesh but a one-dimensional blade discretisation.

We propose two simple comparisons (tables 1 and 2), concerning Agusta aerodynamics department only, with exisiting methods on rotor performance and fuselage pressure distribution prediction.

From these considerations it is therefore clear that Navier-Stokes solution must be used only if it gives something more and not as an alternative.

	Navier-Stokes	Panel/BL
mesh	80-120 h	20 h
CPU time	30-60 h	10-20 min
training	some weeks	some days
reliability no separation	full	full
reliability separation	depends Turb. mod.	depends BL method and coupl.

Table 2: Comparison between Navier-Stokes methods and traditional panel plus boundary layer methods on prediction of pressure distributions over a fuselage. Indication of CPU time is referred to a high capacity Work Station.

5 Industrial requirements

Ideally, a design tool should give reliable and accurate results in a short time, such that modifications can be made and reanalysed without forgetting the previous version. Analysis lasting hundreds of hours on supercomputers are, honestly, unconvenient for design phase.
This not withstanding, Navier-Stokes analysis can be a valuable design tool. If accurate analysis is the objective like the case of fuselage flowfield, particular care must be put in the choice of the solver and the mesh generator.

5.1 Navier-Stokes solver

Speed is essential. The first requirement is therefore the capability to save as much CPU time as possible. As far as steady state flows are simulated, effective acceleration techniques (multigrid, implicit approximated factorization, ...) have to be available and easy to use. For time-accurate calculation the only way is to have an implicit solver, otherwise it is going to take ages.

Beside exact no-slip condition, the solver should allow the user to use and implement his own wall functions boundary conditions. This would give a large saving in CPU time and memory allocation, provided no separation or odd phenomena in the boundary layer are expected.

Compressibility should be accounted for by the solver, although it implies the need for preconditioning in low-velocity zones.

The choice between structured and unstructured approach is free: the first one gives a faster and more accurate code but requires much slower operations of mesh preparation and setting of boundary conditions for each grid block.

5.2 Mesh generation

Hyperbolic algorithms for external grid generation, included in most commercial packages, have proven to be effective. Fully automatic mesh generator for unstructured mesh, are not always reliable and efficient on external geometries.

The preparation of the computational mesh is therefore a lenghty job. We will not give any advice on grid generators but simply note that mesh quality must be compatible with the solver standards.

5.3 Turbulence modelling

The choice of the turbulence model is, from our point of view, of great importance. Fuselage flow has usually separations in the rear part of the cabin, on the fairing and on the tip of the tail cone. Proper simulation of separated zones can be very important for performances and detection of instabilities.

It is well known that separations are not easy to capture properly and that adverse pressure gradient strongly affects turbulence modelling. While algebraic models are not very suitable for complex geometries (for correct definition of "normal wall distance"), in general have low reliability and are thus acceptable for a preliminary analysis only, two-equation models are more general and easy to use.

On the other hand, two-equation models (like the well known $k\epsilon$ and $k\omega$), due to forced isotropy of normal turbulent stresses, hinder separation and usually underestimate separated zones. The best choice would be a non-linear two-equation model or even a second moment closure ($k\omega$ Multiscale or

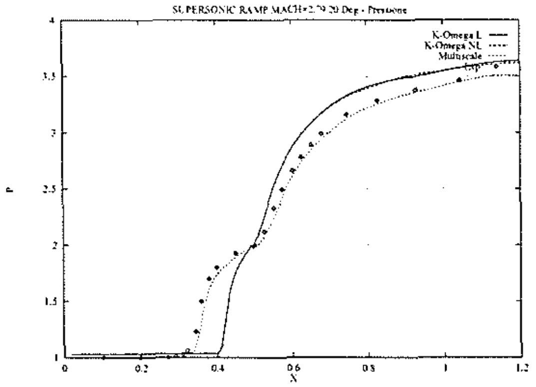

Figure 4: Comparison between turbulence models on a 20° ramp supersonic flow

Launder's proposals), but, of course, it would more CPU-consuming by 30 – 40 %.

A comparison between a two-equation and a 2nd moment closure models is shown in figure 4. Pressure distribution given by a supersonic flow hitting a 20° ramp is strongly affected by an evident separation. The *kω* model underpredicts it while the *kω Multiscale* does not. Most commercial packages do not offer for this kind of model the same level of validation and support. Valuable hints on this topic can be found in [9], [16] and [19, 20].

6 Conclusions

Although the Navier-Stokes solution has become a common and appreciated design tool as far as internal flows are concerned, external flows predictions still have a long way to go.

In helicopter industry internal flow predictions are relevant for applications such as air intakes, exausts or interiors ventilation. Due to flow unsteadiness and complexity, a time and space accurate Navier-Stokes analysis of the flows on rotors or fuselages for engineering purposes (namely reliable predictions and short computation times) is not yet available. Steady fuselage flow is predicted fairly well as the HELIFUSE project has shown, but the uncertainties on predicted drag, the lenghty work necessary for grid preparation, and especially the high sensitivities to grid quality, numerical scheme and turbulence model prevent us, for the time being, from adopting Navier-Stokes analysis as standard design tool for fuselages.

Performance predictions are well obtained with simplified methods for traditional rotors and thus no need for a more expensive tool is felt. As far as unusual innovative configurations are concerned Navier-Stokes methods are not yet ready to give reliable and accurate performance predictions. But they are a powerful analysis tool able to shed light on the flow patterns allowing the designer to achieve a deeper understanding of the phenomena involved, for both existing and innovative configurations. Interaction between rotors and fuselage, identification of large vortical structures, exaust jet trajectory, flow around tail plane at different incidences are just some examples of never well understood flows. It is important to note that as far as these difficult flowfields are concerned, Navier-Stokes analysis, in design phase, has to be coupled at any rate with wind tunnel testing activities in a closed loop interaction.

An accurate flow resolution is not necessarily needed, provided the main characteristics are properly captured. Boundary layer resolution can therefore be simplified with approximate wall functions, for instance, allowing a lower grid resolution which leads to lower aspect ratios and a higher overall grid quality, not to mention the savings in man-hours for grid preparation.

We can draw the conclusion that Navier-Stokes analysis can be a very powerful design tool provided it is used with a pinch of salt, avoiding to spend hundreds of CPU hours trying to compute flow details which would be spoiled by uncertainties due to numerical dissipation, local poor grid quality, time-inaccuracies or badly modelled turbulent phenomena.

References

[1] Amato, M., Boyle, F., Eaton, J., and Gardarein, P., CIRA Technical Report - TR 98072

[2] Berry, J., D., Chaffin, M., S., and Duque, E., P., N., AHS Aeromechanics Specialists Conference, San Francisco, CA, January 1994

[3] Boyd, Jr., D., D., and Barnwell, R., W., AHS 54th Annual Forum, Washington, D. C., May 20-22, 1998

[4] Buchtala, B., Wehr, D., and Wagner, S., ERF 1997

[5] Costes, M., et al., AHS 54th Annual Forum, Washington, D. C., May 20-22, 1998

[6] Dindar, M., Lemnios, A., Shepard, M., Jansen, K., and Kenwright, D., AHS 54th Annual Forum, Washington, D. C., May 20-22, 1998

[7] Duque, E., P., N., and Dimanlig, A., C., B., AHS Aeromechanics Specialists Conference, San Francisco, CA, January 1994

[8] Fejtek, I., and Roberts, L., AIAA 91-0707, Presented at the AIAA 29th Aerospace Science Meeting, Reno, Jan. 1991

[9] Lien, F. S., and Leschziner, M. A., Computers Fluids Vol. 23, No. 8, 1994

[10] Meakin, R., L., AIAA 93-4878, Presented at the AIAA Atmospheric Flight Mechanics Conference, Baltimore, Aug 1995

[11] Narramore, J., C., and Brand, A., G., AHS 48th Annual Forum, Washington, D. C., June 1992

[12] Narramore, J., C., AHS Aeromechanics Specialists Conference, San Francisco, CA, January 1994

[13] Poling, D., R., Rosenstein, H., and Rajagopalan, R., G., AHS Journal, April 1998

[14] Rajagopalan, R., G., and Mathur, S., R., AHS Journal, July, 1993

[15] Rajagopalan, R., G., and keys, C., N., AHS Journal, October 1997

[16] Righi, M., PHD Thesis, *Politecnico di Milano*, Milan, May, 1998

[17] Srinivasan, G., R., Raghavan, V., Duque, E., P., N., and McCroskey, W., J., AHS Journal, July, 1993

[18] Wake, B., E., and Baeder, J., D., AHS Journal, January 1996

[19] Wilcox, D. C., AIAA Journal 31 (1993)

[20] Wilcox, D. C., DCW Industries Inc., 1993

[21] Zori, L., A., J., and Rajagopalan, R. G., AHS Journal, April 1995