Back to Session Subjects

Development of a Wall Treatment for Navier-Stokes Computations using the Overset Grid Technique

Th. Schwarz
DLR, Institute of Design Aerodynamics
Lilienthalplatz 7, 38108 Braunschweig, Germany

This paper presents a method to perform Navier-Stokes computations with the overset grid (or Chimera) approach with grid overlap on body surfaces. The method allows a correct transfer of the flow variables in the boundary layer near curved surfaces. Without the correction the flow is disturbed at the Chimera boundaries.

The functionality of the new method is demonstrated for a simple 2D NACA 0012 test case and a more complex 3D helicopter fuselage with wind tunnel support strut. The results for the helicopter configuration are in good agreement with measurements and a computation with a standard multiblock grid.

1. Introduction

The overset grid technique (also known as Chimera grid technique) has been proven to be a reliable tool to compute the flow around complex configurations. This method introduced by Benek and Steger [1] allows to subdivide the physical domain into separate regions and to generate a mesh for each region. In contrast to the multiblock approach, the meshes overlap at the grid boundaries. If some grid points lie inside a solid body, these nondiscretizable grid points are excluded from the flow computation. The removal of points is called hole cutting. At the Chimera boundaries and hole fringe points flow data are transferred from an overlapping donor grid. This intergrid communication is usually established by interpolation techniques.

Most of the current Chimera computations are performed for bodies in relative motion, for example, store separation or helicopter configurations [2]-[6]. A second way to use the Chimera technique is to make the grid generation process easier. This approach allows to mesh complicated configurations with a set of relatively simple overlapping body-fitted grids [7]. In this paper the second approach will be followed.

The application of the Chimera method in general is easy. But care must be taken if a grid overlap exists on a body surface especially for Navier-Stokes flow computations. With a standard implementation of the Chimera technique the boundary layer is strongly disturbed at Chimera grid boundaries. In order to overcome this problem a method was developed within the French-German project CHANCE (Complete Helicopter AdvaNced Computational Environment) which allows the correct transfer of flow variables.

The paper is structured as follows: First, the numerical method is presented which was used to compute the results shown in this paper. In the third chapter the problem of flow disturbances at Chimera boundaries is described and the method to overcome this problem is presented. In the fourth chapter, the new method is applied to a helicopter fuselage with wind tunnel model support strut. The results will be compared with experimental data and standard multiblock computations obtained during the BRITE-EURAM HELIFUSE project.

2. Numerical method

2.1 Basic solution algorithm

All computations were carried out using the DLR flow solver FLOWer [8]. FLOWer solves the unsteady, compressible three dimensional Reynolds averaged Navier-Stokes equations on block-structured meshes. The approximation of the governing equations follows the method of lines, which decouples the discretization of space and time. The spatial discretization is based on a finite volume method, where the flow variables are calculated at the vertices of the grid cells. The control volume is build by the eight cells surrounding the respective grid node. The fluxes through the cell faces are approximated using a central discretization operator. Since the finite volume discretization based on central averaging is not dissipative high frequencies are not damped. In order to avoid these spurious oscillations, a blend of second and fourth differences is implemented according to

Jameson [9] with modifications by Martinelli [10].

For the integration in time, an explicit five stage Runge-Kutta time stepping scheme is used.

The convergence can be accelerated by implicit residual smoothing, local time steps and multigrid.

2.2 Turbulence modeling

In the FLOWer code, turbulence is modeled either by the algebraic model of Baldwin-Lomax or by more general one or two equation turbulence models. In this study, the Baldwin-Lomax model and the LEA k-ω model is used (LEA = Linearized Explicit Algebraic stress) [11]. The LEA k-ω model represents the linear part of a non-linear explicit algebraic stress model written in terms of the Wilcox k-ω formulation. It combines the advantages of Reynolds stress modeling accuracy with the numerical advantages of the eddy viscosity concept.

2.3 Preconditioning

For small Mach numbers the classical methods used to solve the compressible Navier-Stokes equations may give poor results since the governing equations become numerically stiff. In order to overcome this difficulty, a preconditioner, proposed by Choi and Merkle [12][13], is implemented into FLOWer.

2.4 Chimera method

FLOWer has full Chimera functionality [14]. The method allows for arbitrary hierarchies of grids: Every grid may overlap with an arbitrary number of other grids which may overlap themselves.

The transfer of flow data at Chimera boundaries is performed by trilinear interpolation. In order to determine the donor cell and the interpolation coefficients for a given target point, first the donor grid is searched for possible donor cells with an alternating digital tree search algorithm (ADT) [15]. The ADT gives the indices of one or more cells which must be checked more accurately. For this, each cell is subdivided into six non overlapping tetrahedrons and it is tested which tetrahedron encloses the target point. This tetrahedron is used to calculate the interpolation coefficients.

In order to define „holes“ in a grid all points lying inside a user specified volume are blanked. The volumes may be boxes, cylinders, grids etc. .

3. Overlapping surface grids

The Chimera method as implemented into FLOWer gives good results if the Chimera boundaries are away from body surfaces. But if a grid overlap exists on a body surface the flow inside the boundary layer is disturbed at the Chimera mesh boundaries. An example is given in figure 1: It shows a 2D Navier-Stokes grid around a NACA 0012 airfoil. Near the leading edge, all grid points lying inside a rectangular box are ‚blanked‘ and are therefore not used for the flow calculation. The blanked region is covered with a small body-fitting grid. The resulting pressure and friction distributions for the airfoil at Ma=0.8 and $\alpha=1.25^{o}$ using the Baldwin-Lomax turbulence model are shown in figure 2. While the pressure distribution is as expected, the friction distribution exhibits very high values in the overlap region. Further investigations with other test cases show, that this problem does always occur for overlapping grids on curved surfaces. The magnitude of the error depends on the curvature and the cell aspect ratio of the surface grids.

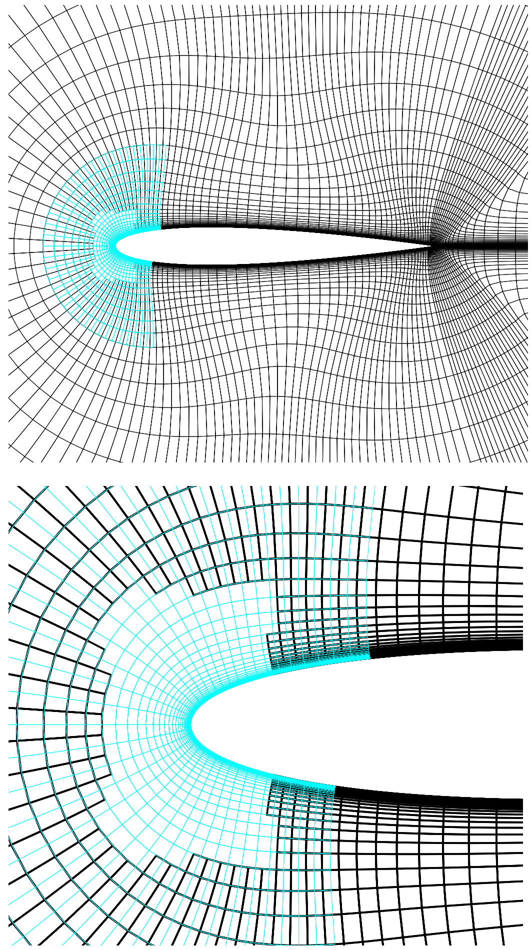

Figure 1: top: Chimera grid around NACA 0012 airfoil;bottom: detailed view

The reason for this phenomenon is the different discretization of the body surface with the overlapping grids. This results in a wrong transfer of the flow variables at the Chimera grid boundaries. For the following explanation a spatial discretization with the

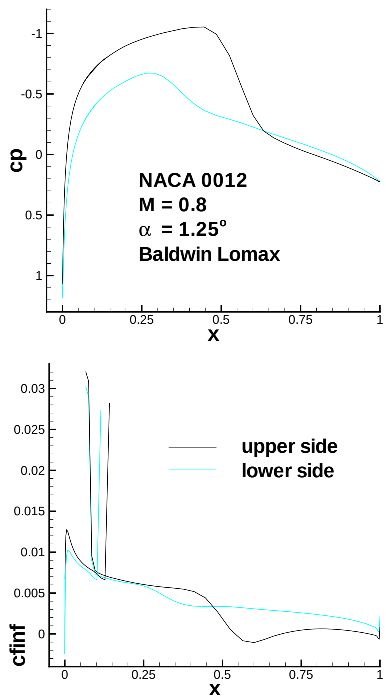

Figure 2: Pressure and skin friction distribution without wall correction

flow variables being discretizied at the nodes of a grid is assumed. Similar considerations are valid for a cell centered discretization.

Depending on the shape of the body surface, two different cases occur: Figure 3 shows an overlapping grid near a concave surface. It is evident that one

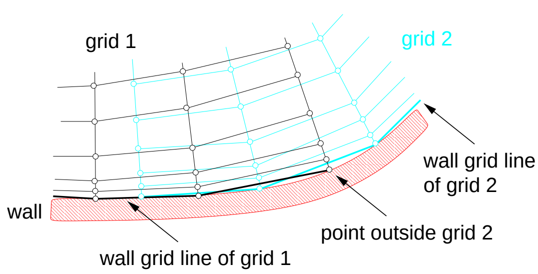

Figure 3: Grids near concave surface

point of grid 1 is outside of grid 2. This is caused by the straight grid lines between the nodes of grid 2. An accurate interpolation of flow variables for this grid node is therefore impossible. It is clear, that for a higher surface curvature or a higher cell aspect ratio even more grid nodes of grid 1 may be outside of grid 2. In the case of a convex surface, see figure 4. all grid nodes of grid 1 are inside of grid 2. The interpolation of flow variables is therefore possible. Nevertheless, the interpolation coefficients are still incorrect, since the surface grid nodes of grid 1 have a certain distance to the wall grid line of grid 2. In figure 5, top this distance is marked with $\vec{\delta}$. With a standard implementation of the Chimera method the

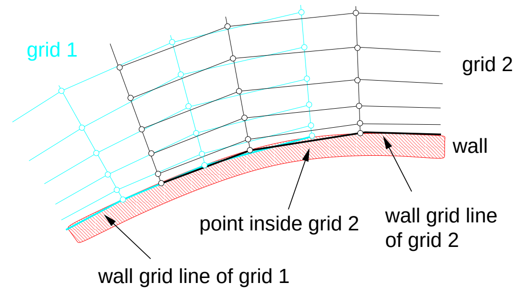

Figure 4: Grids near convex surface

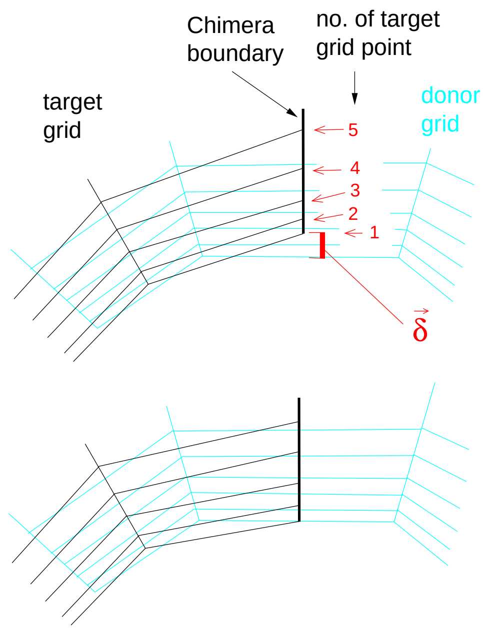

Figure 5: Principle of wall correction, top: original grids; bottom: modified target grid

flow variables are interpolated from the donor mesh at the physical locations of the target grid points 'i'. Concerning the distance to the wall this introduces an error in location of $\vec{\delta}$ and results in a wrong transfer of the flow variables. It is evident that this error will vanish for an infinite refined mesh.

Remembering the given NACA 0012 example, the velocity is highly varying inside the boundary layer. Therefore, small errors in the location of flow transfer result in high errors in the transferred velocity profile which implicate the observed wrong values for the skin friction (figure 3). In contrast to the velocity, the pressure is almost constant inside a boundary layer. Therefore, the pressure distribution is not influenced by this flaw.

In order to overcome the difficulties, two methods are possible: First the interpolation coefficients can be calculated with the grid generator since it uses the exact geometry of a configuration. This requires a calculation of the coefficients based on cells with curved edges. The disadvantage of this approach is that for a given grid the original surface geometry is possibly not available. Furthermore it requires the coupling of the grid generator with the flow solver for unsteady computations. Due to this reasons a second method was chosen to compute correct interpolation coefficients. It uses only the grid coordinates of a given grid.

The principle approach with the new method is to modify the target grid in a way that the Chimera points lying at a wall are shifted to the wall grid lines of the donor grid, see figure 5. Then the standard interpolation scheme can be used to derive the coefficients. Afterwards, the regridded target grid is not needed anymore since the flow computation still uses the original grids.

The algorithm for this procedure in 2D is:

1. Create a copy of the original target grid.
2. Find a Chimera point of the target grid which lies on a wall (point ‘1‘ in figure 5).
3. Construct a line normal to the wall and through the target point.
4. Compute the coordinates $\vec{X}_{intersect}$ of the intersection of the line with the wall grid line of the donor mesh. This involves a searching process for the matching wall cell of the donor mesh.
5. Compute a vector from the target point ‘1‘ to the intersection coordinates

$$\vec{\delta} = \vec{x}_{intersect} - \vec{x}_1 \tag{1}$$

6. Take the grid line of the target grid, which starts at the target node and is normal to the wall, and compute the arclength from the target point to the other points of the grid line (points ‘i‘ in figure 5).

$$l_i = \sum_{k=1}^{i-1} \left|\vec{x}_{k+1} - \vec{x}_k\right| \tag{2}$$

7. Shift the target grid node of the copied target mesh to the wall grid line of the donor mesh. Shift the nodes of the grid line normal to the wall also in the direction of the donor wall grid line, but with a decreasing shift distance for an increasing distance to the wall

$$\vec{x}_{i,\,new} = \vec{x}_i + \alpha \cdot \vec{\delta} \tag{3}$$

$$0 \le \alpha \le 1 \tag{4}$$

   The weighting function α depends on the arclength l_i which was calculated with equation (2):

$$\alpha = f(l_i) = \begin{cases} 1 \;\; at\ wall \\ 0 \;\; afar\ \ from\ wall \end{cases} \tag{5}$$

8. Repeat step 2) to 7) for all Chimera points lying at a wall.
9. Use the modified target grid for the computation of the interpolation coefficients.

The extension of this method for a three dimensional grid is straight forward.

With this wall correction the boundary layer data are transferred correctly at the Chimera grid boundaries. The computed friction distribution for the NACA 0012 airfoil is now as expected, see figure 6.

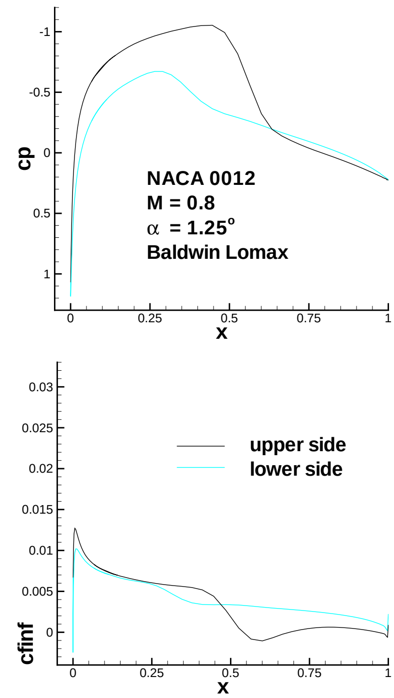

Figure 6: Pressure and skin friction distribution with wall correction

4. Application to helicopter fuselage

This method is also applied to a flow computation around the EUROCOPTER DGV fuselage. The results are compared to measurements and computations performed within the BRITE-EURAM project HELIFUSE [16][17]. One experimental setup was a simplified fuselage which was mounted on a model support strut for wind tunnel testing. Previous investigations have shown, that the fuselage drag can be predicted accurately only, if the strut effects are taken into account [17]. While in [17] a standard multiblock mesh was used for the fuselage-strut configuration, for this paper the Chimera approach is pursued. The grid system consists of a three block grid around the fuselage which is overset with a grid around the strut, see figure 7. Figure 8 shows a detailed view of the overlap region at the surface.

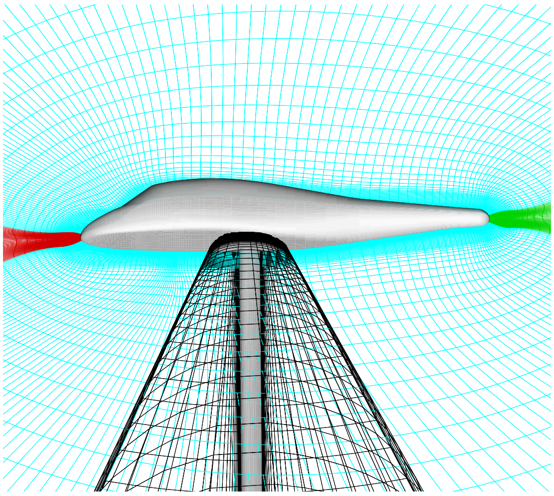

Figure 7: Grid setup for HELIFUSE Configuration

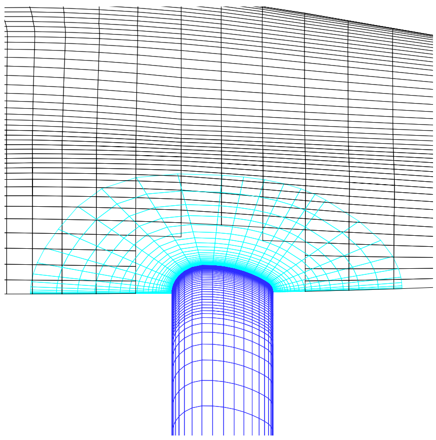

Figure 8: Overlap at wall and hole for fuselage and strut grids

The flow is computed with FLOWer for the Mach number 0.235 and the Reynolds number $30*10^6$. These conditions correspond to the HELIFUSE test case TS2. Two flow calculations are carried out: For both preconditioning and the two equation LEA-k-ω turbulence model is used, but one is performed without any special wall treatment, while the other uses the wall correction as described above.

The computation without wall treatment shows a highly disturbed flow at the grid overlap region (see figure 9) whereas the computation with wall correction gives proper surface stream lines as shown in figure 10. The iso-friction lines are also continuous across the overlap region, see figure 11.

Direct integration of the flow quantities in order to compute the global forces (lift, drag, moments) would count the forces in the overlapping regions more than once. Therefore, a postprocessing tool was developed which reads in the coordinates of the body surface and the corresponding flow variables. Then the tool removes the grid overlap. This introduces a

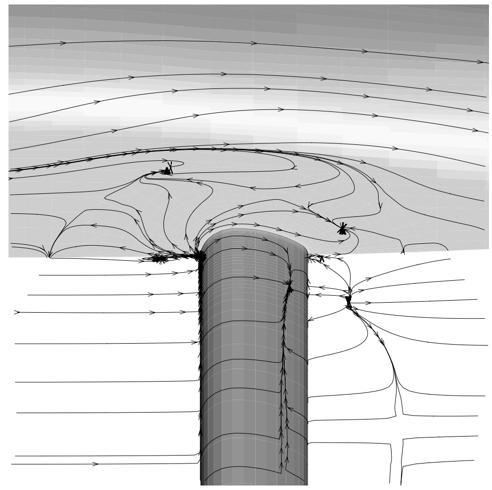

Figure 9: Streamlines without correction

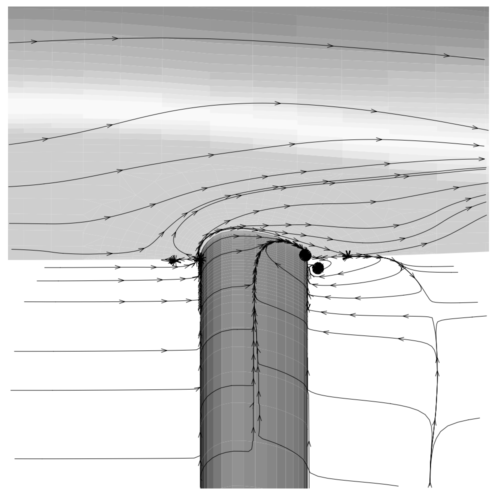

Figure 10: Streamlines including correction

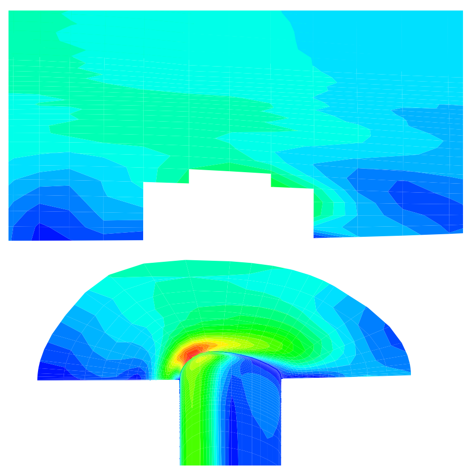

Figure 11: Local friction at surface including correction

gap between the grids which is filled with triangles as presented in figure 12. The resulting grid covers the body surface only once and allows to compute the global forces. A similar tool was developed by Chan and Buning [18].

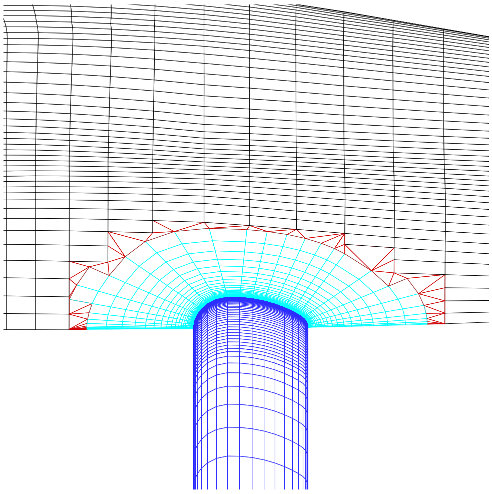

Figure 12: Grid overlap removed for global force computation

Figure 13 demonstrates the good congruence of the computed drag with the experiment and the conventional multiblock computation. Compared to the results for the bare fuselage the influence of the strut on the drag prediction is also evident.

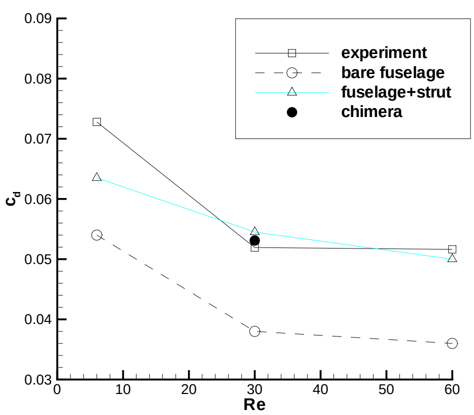

Figure 13: Comparison of drag prediction

5. Conclusion

A method has been developed which allows the correct computation of the viscous flow for Chimera computations with overlapping grids on curved body surfaces. The algorithm has been implemented into the FLOWer flow solver. The functionality was demonstrated for a NACA 0012 airfoil. The method has also been successfully applied to a helicopter fuselage with wind tunnel support strut. The results for the computations with LEA-k-ω turbulence model are in very good congruence with experimental data and results obtained with a standard multiblock computation.

References

[1] Benek, J.A.; Buning, P.G.; Steger, J.L.: *“A 3-D Chimera Grid Embedding Technique“*, AIAA 7th Computational Fluid Dynamics Conference, Paper AIAA-85-1523, July 1985, Cincinnati, USA

[2] Gillyboef, J.P.; Mansuy, P.; Pavsic, S.: *“Two New Chimera Methods: Application to Missile Separation“*, 33rd Aerospace Sciences Meeting and Exhibit, January 1995, Reno, NV, USA

[3] Pahlke, K.; Boniface, J.-C.: *“A Detailed Comparison of DLR and ONERA 3D Euler Methods for Rotors in High Speed forward flight“*, 24th European Rotorcraft Forum, Paper AE14, September 1998, Marseilles, France

[4] Stangel, R.; Wagner, S. : *“Euler Simulation of a Helicopter Configuration in Forward Flight using a Chimera Technique“*, 52nd Annual Forum of the American Helicopter Society, June 1996, Washington, D.C., USA

[5] Boniface, J.-Ch.; Guillen, Ph.; Le Pape, M.-C.; Darracq, D.; Beaumier, Ph. : *“Development of a Chimera Unsteady Method for the Numerical Simulation of Rotorcraft Flowfields“*, 36th AIAA Aerospace Science Meeting & Exhibit, Reno, NV(USA), January 1998

[6] Meakin, R.: *“Moving Body Overset Grid Methods for Complete Aircraft Tiltrotor Simulations“*, paper AIAA-93-3350-CP

[7] Meakin, R.L.; Wissink, A.M.:*“Unsteady Aerodynamic Simulation of Static and Moving Bodies using Scalable Computers“*, 14th AIAA Computational Fluid Dynamics Conference, Paper AIAA-99-3302, June 1999, Norfolk, VA, USA

[8] Kroll, N.; Rossow, C.C.; Becker, K.; Thiele, F.:*“MEGAFLOW-A Numerical Flow Simulation system“*, ICAS-congress, September 1998, Melbourne, Australia

[9] Jameson, A.; Schmidt, W.; Turkel, E.: *“Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes“*, AIAA Paper 81-1259, 1981

[10] Martinelli, L.; Jameson, A.: *“Validation of a Multigrid Method for the Reynolds-Averaged Navier-Stokes Equations“*, AIAA Paper 88-0414, 1988

[11] Rung, T.; Lubcke, H.; Franke, M.; Xue, L.; Thiele, F.; Fu, S.: *“Assessment of Explicit Algebraic Stress Models in Transonic Flows“*, Sym-

posium on Engineering Turbulence Modeling and Measurements, Corsica, France, 1999

[12] Choi, Y.H.; Merkle, C.L.: *"The Application of Preconditioning to Viscous Flows"*, Journal of Computational Physics, Vol. 105, pp 207-223, 1993

[13] Turkel, E.; Radespiel, R.; Kroll, N.: "Assessment of Two Preconditioning Methods for Aerodynamic Problems", Computers and Fluids, Vol. 26, pp 613-634, 1997

[14] Pahlke,K.: *"Berechnung von Strömungsfeldern um Hubschrauberrotoren im Vorwärtsflug durch die Lösung der Euler-Gleichungen"*, DLR-Forschungsbericht 1999-22, ISSN 1434-8454, 1999

[15] Bonet, J.; Peraire, J.: *"An Alternating Digital Tree (ADT) Algorithm for 3D Geometric searching and Intersection Problems"*, International Journal for Numerical Methods in Engineering, Vol. 31, pp 1-17, 1991

[16] Gatard, J.; Costes, M.; Kroll, N.; Renzoni, P.; Kokkalis, A.;Rochetto, A.; Serr, C.; Larry, E.; Filippone, A.; Wehr, D.: *"High Reynolds Number Helicopter Fuselage Test in the ONERA F1 Pressurized Wind-Tunnel"*, 23rd European Rotorcraft Forum, 1997, Dresden, Germany

[17] von Geyr, H.; Kroll, N.: *"Application of 3D-Preconditioning for the prediction of Helicopter Fuselage Drag considering interferences with model support strut"* , 25th European Rotorcraft Forum, Paper C10, September 1999, Rome, Italy

[18] Chan W.M. ; Buning P.G.: *"Zipper Grids for Force and Moment Computation on Overset Grids"*, AIAA Paper 95-1681-CP