SYNTHESIS OF ONERA CHIMERA METHOD
DEVELOPED IN THE FRAME OF CHANCE PROGRAM §

C. Benoit
Research Scientist
ONERA,
Christophe.Benoit@onera.fr
Châtillon, FRANCE

G. Jeanfaivre
Research Scientist
ONERA,
Gaelle.Jeanfaivre@onera.fr
Châtillon, FRANCE

E. Canonne
Research Scientist
ONERA,
Elodie.Canonne@onera.fr
Châtillon, FRANCE

Abstract

Within the frame of the long term French-German CHANCE program, the Chimera method has been developed and enhanced particularly for helicopter applications. This paper intends to give a review of the different improvements that have been carried out along those six years, insisting on the gain that they individually brought, enabling finally complete helicopter flow simulations.

1 Introduction

Nowadays, Computational Fluid Dynamics has come to a mature point. Numerical Simulations around complex geometries with moving parts have become not only possible, but also efficient. Codes dealing with this kind of applications make often use of the Chimera method [3]. Major codes based on this technique are NASA's Overflow [13], Beggar [16] and JAXA's code [1]. In the frame of the CHANCE project, the Chimera method has been developed both on the German side in the Flower solver and on the French side in the elsA solver.

There was three axis in our work. The first one was adapting Chimera to complex geometries, the second one was efficiency and the last one was using Chimera for mesh adaptation.

In the first part, our idea was to improve the applicability domain of Chimera by alleviating the overlap constraint and enabling a large possibility of mesh components associations [9]. In the second part, mainly two methods have been studied: multigrid/Chimera [11] and parallel Chimera [10]. In the last axis of our work, two adaptation techniques based on Chimera have been developed. The first one, based on the work of Meakin [14] consists in automatic generation and adaptation of cartesian background grids. It is quite general and can be applied to a lot of applications [4]. The second one is based on the use of a cylindrical grid as a background grid and is dedicated to rotors in hover [5].

§Presented at the 31st ERF Forum,
Florence, Italia, September 13-15, 2005.

2 Chimera for complex configurations. The mesh components approach.

2.1 Basic Chimera method

The purpose of Chimera is to solve a set of time marching equations on a set of overset grids. In our case, it is used to solve the compressible Euler or Reynolds average Navier-Stokes equations on a set of overset structured grids. The equations are solved on each grid in the finite-volume framework, using the Jameson scheme [8] and the implicit residual smoothing implicit phase [12] or the LDU implicit phase [21].

Basically, Chimera technique consists in two kind of intergrid transfers: in overlap boundaries and around blanking regions (see Fig. 1). Overlap boundaries concern the outer boundary of overset grids (in green on the figure), solution is interpolated on the two layers of tagged points. So, the classical numerical scheme can be applied on the other points of the domain. The blanking region is made of points that lie inside the solid body. Interpolated points are fringed around the holes (in red on the figure). Blanked points are not computed and are uncoupled in the implicit phase.

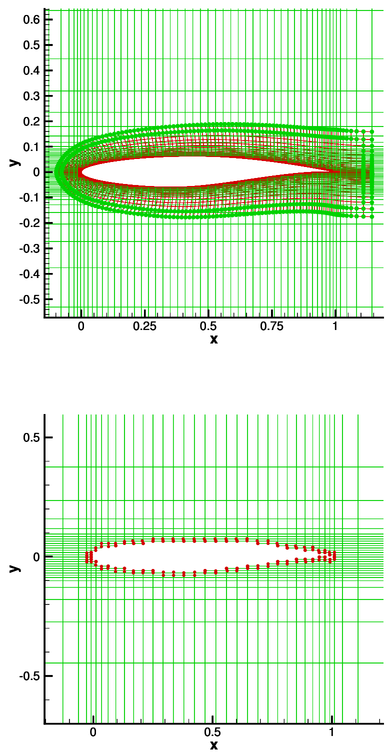

Figure 1: Chimera intergrid communications.

For the intergrid communications to work properly, a sufficient overlap must exists between the grids, such that interpolated points can be interpolated from a valid cell (see Fig. 2). This necessary minimum overlap is really a problem since it results in constraints in the mesh generation. It can force the user to redo its mesh after having tested Chimera connectivity and may be a jigsaw especially for moving bodies applications.

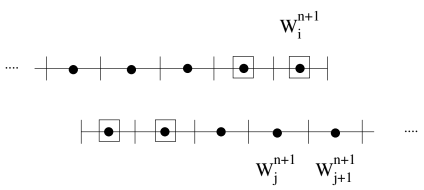

Figure 2: Correct overlap in one space-dimension for two layers of interpolation cells. Interpolated points are tagged with a square.

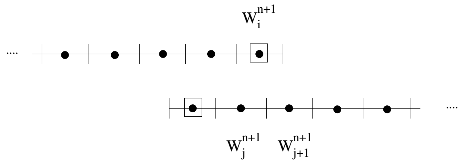

Figure 3: Correct overlap in one space-dimension for one layer of interpolation cell.

2.2 One layer of interpolated cells

To reduce the occurrence of minimum overlap problems, we have enable Chimera transfers to be performed only on one layer of interpolated points. This allows a really shorter overlap (see Fig. 3). Of course, the numerical scheme is modified in the neighboring cell. An interpolated point is computed in the cell interface center and used in the modified numerical scheme. The overall scheme is second order accurate.

2.3 Implicit interpolation

To enable even smaller overlaps, implicit interpolation has been developed. For instance on Fig. 4, one interpolated point has an interpolation containing itself an interpolated point. This problem can nevertheless be solved by writing interpolation relationships at time n+1. Some values are already known by the numerical schemes, some are unknown. By writing and inverting this system, one can get the interpolated value at time n+1.

At this point, Chimera can be seen as an easy way of adding new body components to a CFD problem, when they are not in contact with or intersecting other bodies. For instance, blades can be added to a fuselage mesh or a missile can be added under a wing mesh. Of course, when bodies are in contact or intersecting, no minimum overlap can be found. Techniques to solve this kind of problems will be presented in next sections.

Together with Onera's Applied Aerodynamics team, those techniques has been used to compute helicopter rotor-fuselage applications for Euler and viscous flows [17]. In particular, the introduction of one layer of interpolation cells makes the computation of realistic rotors near the fuselage possible (see Fig. 5). The presented configuration is a simplified Dauphin fuselage with a 7A model rotor ($M_{tip} = 0.646, \mu = 0.4$) and is computed by solving the Euler equations..

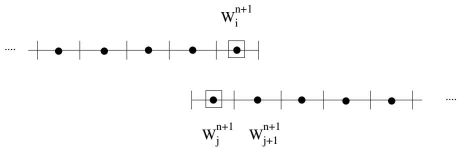

Figure 4: Correct overlap in one space-dimension for one layer of interpolation cell and implicit interpolation.

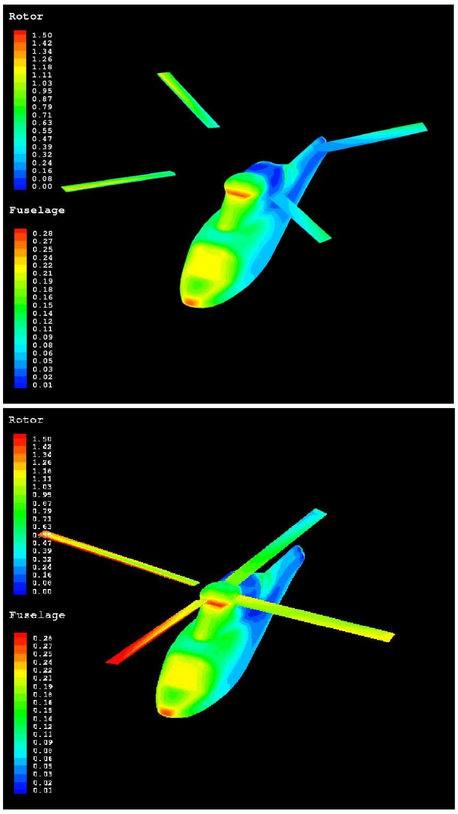

Figure 5: One layer of interpolation cells enables rotor to be placed nearer of fuselage.

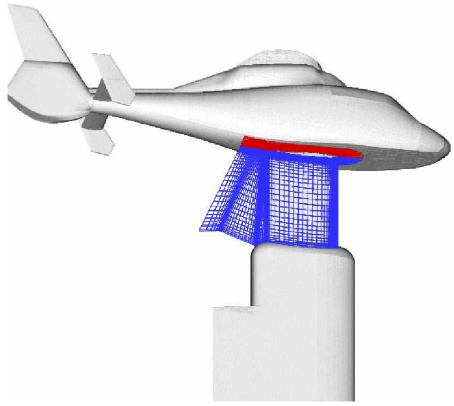

Figure 6: Strut added under a fuselage using multiply defined walls technique.

2.4 Multiply defined walls

To be a used as a real design tool, Chimera must furnish a way of adding bodies or part of bodies in contact with another one. The purpose of it being, for example, to design a simple fuselage mesh and then to add stabilizers, landing gears,... first to simplify mesh generation and enable mesh reusability and then to be able to measure the impact of each component on the overall flow without remeshing effort.

One way of adding a body in contact with another one using Chimera is to have the mesh of one body conforming to the other body surface. For instance, on Fig. 6, a strut has been added to an isolated fuselage mesh. The mesh of the strut is conforming to the fuselage mesh on the upper surface. This way, no blanking is necessary in the strut mesh, making it valid for interpolation for all blanked points in the fuselage mesh. Nevertheless, due to the discretization discrepancies, a correction of interpolation coefficients must be performed for the interpolated points near the surface [20] in particular to compute an accurate friction coefficient.

This technique has been used to compute fuselage and strut, fuselage and stabilizers in [18].

2.5 Gridless solver

The previous technique lacks of generality in the sense that one body mesh must be adapted to the other body. To alleviate that and enable absolute general mesh components association, a specific technique has been developed. It is based on the first observation that, in the case of general intersection of bodies with

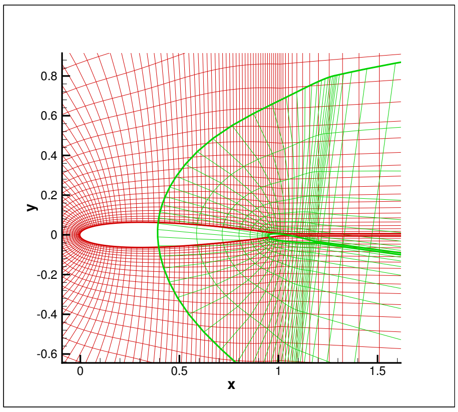

Figure 7: Arbitrary intersecting meshes around profile and flap.

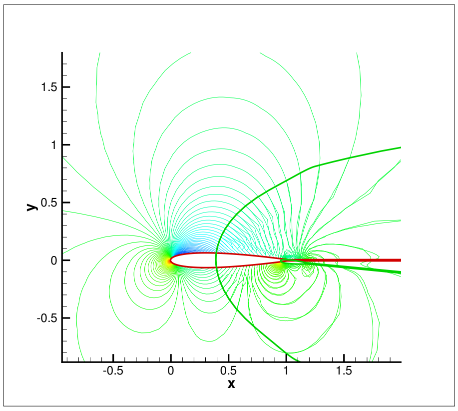

Figure 8: Flow around two intersecting bodies using the Gridless technique. Iso-density lines.

Chimera, a small region of the computational domain near the intersection can not be nor discretised nor interpolated. In this region, we propose to use a grid-less solver [2] [7], written on a cloud of points, that gathers local valid points of different grids and created points on the wall. This technique is just at its premiss and gives only positive results in 2D and for the Euler equations [15]. Nevertheless, we are still working on that.

For instance, the inviscid flow simulated around a NACA0012 profile and an intersecting flap with 6 degrees of incidence at free-stream Mach number of 0.4 is presented here. The mesh simply consists in a monoblock mesh around the profile overset with a monoblock mesh around the flap (Fig. 7). Iso-density lines around the complete configuration are shown on Fig. 8.

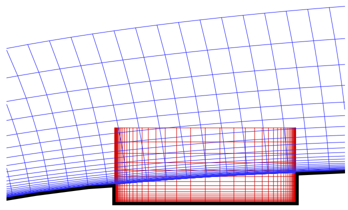

Figure 9: A surface (in bold black) defined by two grids.

2.6 Doubly defined boundary conditions

Another way to define a composite body surface is to make holes in a given surface. Fig. 9 shows a surface defined this way: the first surface is digged by an overset grid, the resulting composite surface is displayed in bold black. To enable easily this kind of association, a user facility has been added to standard Chimera boundary conditions: a boundary condition can be defined by a classical boundary condition (wall, non reflection, ...) and also by an overlap boundary condition. If a point of a boundary can be interpolated, it will be. Otherwise, the classical boundary condition is used. Using this technique in association with one layer interpolated points and implicit interpolation enables the easy definition of cavities, slots for cooling jets, etc...

For instance, the viscous flow simulated around a cavity in a NACA0012 profile is presented now. The mesh is made of three grids. One for the close profile, a cartesian grid as a background grid and a cartesian grid for the cavity (see Fig. 10). The free-stream mach number is 0.73 and the Reynolds number is 6.5 10^6. The solution is shown on Fig. 11.

2.7 Deforming bodies

ALE method has been independently developed in the CHANCE program [6]. Then, Chimera method can be used together with the ALE formulation, enabling for example the computation of elastic blades. It has been used for instance in [19] to compute an elastic rotor above a wind tunnel model support.

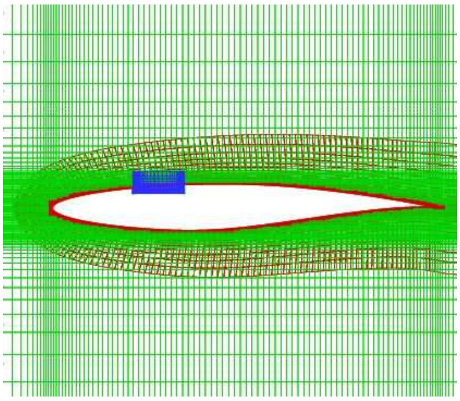

Figure 10: Mesh for a cavity defined by a doubly defined boundary condition.

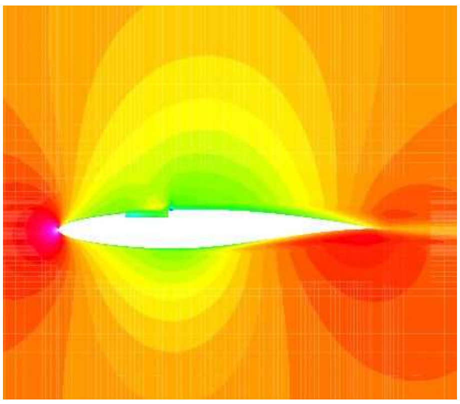

Figure 11: Iso-density lines for a cavity defined by a doubly defined boundary condition.

3 Efficiency of Chimera

3.1 Chimera and multigrid

Multigrid has been made compatible with Chimera. The main problem was the overlap between coarse grids that is nearly everywhere insufficient. Boundary conditions for coarse grids based on extrapolation or interpolation using the fine grid have been developed and were demonstrated to perform correctly [11].

3.2 Parallel Chimera

The interpolation cell search and the Chimera transfers work in parallel using distributed memory and MPI communications. Care has been taken to minimize the amount of data in the transfers. For instance, interpolation coefficients are stored in the interpolation block, such that only the final interpolated field must be exchanged. Reported speed-up were about 3 on four processors and 6 on height processors for steady applications and 2.5 for four processors and 3.5 for eight processors for unsteady applications. A set of results is presented in [10].

4 Automatic mesh generation and adaptation based on Chimera

4.1 Cartesian mesh generation and adaptation based on Chimera

In this method, the bodies are described by short grids around the body surface. The previous techniques can be used to define those body grids. From this body grids, a set of automatically generated cartesian grids is produced (Fig. 12). Their step is first computed from a proximity rule to the body grids and then also based on a local refinement indicator. The connectivity between cartesian grids can be based on Chimera or on patch grid joins. It has to be noticed that a correct overlap between body grids and cartesian grids is automatically ensured [4].

This method is the ideal complement of the previous mesh component association techniques, enabling an easy definition of the remaining part of the computational domain and an accurate capture of the flow details.

The key points here is the efficiency of cartesian grids: solver on cartesian grid can be simplified and

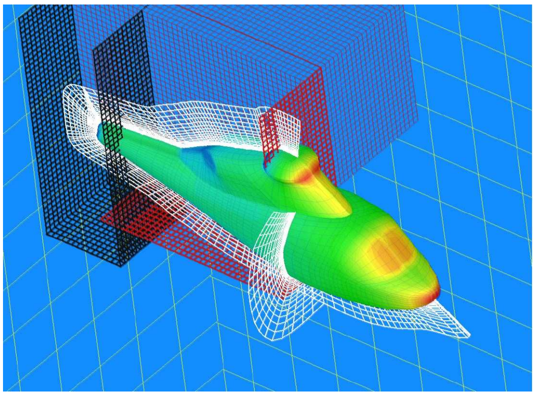

Figure 12: An automatically generated cartesian mesh around a fuselage.

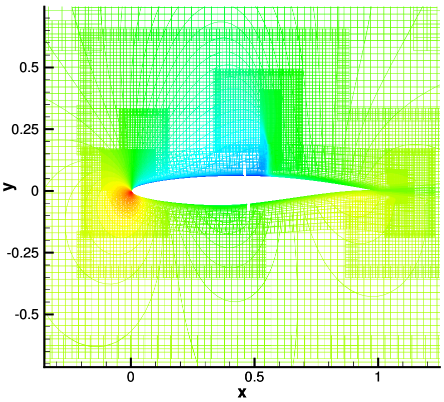

Figure 13: An automatically generated cartesian mesh around a RAE2822 profile. Iso-density lines.

metrics may not be stored. To our experience, 10 % CPU time and 20 % memory can be saved compared to the standard structured curvilinear solver. Besides, high accurate numerical schemes can be easily implemented on those grids.

The turbulent flow around a RAE2822 profile at $M_\infty = 0.76$ and Re $= 6.5\ 10^6$ is presented on Fig. 13 showing a clear adaptation in the shock wave region and in the wake. This method has been also used to compute rotors in hover, rotors in forward flight and isolated fuselages.

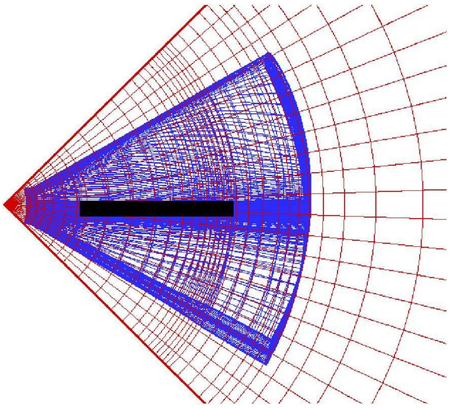

Figure 14: An automatically generated cylindrical grid and the blade grid.

Figure 15: Iso-vorticity surface obtained after mesh adaptation.

4.2 Cylindrical mesh adaptation for rotors in hover

For the purpose of computing rotors in hover, a specific technique has been developed. For this kind of application, a cylindrical topology seems to be well suited since, besides its simplicity, it grossly follows the tip vortex path. In our technique [5], the user must provide a short grid around the blade, then a cylindrical grid is automatically generated that is refined in the wake. Both grids are overset and Chimera transfers are used between the two grids. The numerical scheme has also been improved and is now able to compute accurately irregular cylindrical meshes.

The mesh obtained for 7A rotor in hover after 6 remeshing is shown on Fig. 14. The tip vortex can be followed for about 380 degrees of age (Fig. 15).

4.3 Conclusion

Nowadays, Chimera seems compulsory for dealing with helicopter applications where moving bodies are concerned. In this project, we have focussed on improving its applicability and efficiency. For a lot of helicopter applications, the method has proven its usefulness. Of course, some of the techniques presented here are always under investigation and need further developments to be used in industrial environment.

4.4 Acknowledgments

Authors would like to thank all the people involved in the CHANCE project, especially Philippe Beaumier, Jean-Christophe Boniface, Sylvie Borie, Bernard Cantaloube, Christophe Castellin, Michel Costes, Alexandro D'Alascio, Vincent Gleize, Klaus-Dieter Pahlke, Frédéric Le Chuiton, Walid Khier, Thomas Renaud, Benoit Rodriguez, Thorsten Schwarz, Jacques Sidès,... and the French DGA and DPAC for funding.

References

[1] T. Aoyama, C. Yang, and S. Saito. Numerical analysis of active flap for noise reduction using moving overlapped grid method. In *proceedings of the* 61^{st} *AHS Annual Forum, Grapevine*, 2005.

[2] J.T. Batina. A Gridless Euler-Navier-Stokes Solution Algorithm for Complex Aircraft Applications. In *proceedings of the* 31^{st} *AIAA Aerospace Sciences Meeting and Exhibit, Reno*, AIAA 93-0333, 1993.

[3] J.A. Benek, J.L. Steger, and F.C. Dougherty. A Flexible Grid Embedding Technique with Application to the Euler Equations. AIAA Paper 83-1944, January 1983.

[4] C. Benoit and G. Jeanfaivre. 3D Inviscid Rotor and Fuselage Calculations Using Chimera and Automatic Cartesian Partitioning Methods. In *Journal of the AHS*, vol. 48, No 2, April 2003.

[5] E. Canonne, C. Benoit, and G. Jeanfaivre. Cylindrical mesh adaptation for isolated rotors in hover. In *proceedings of the* 58^{th} *AHS Annual Forum, Montreal*, June 2002.

[6] B. Cantaloube and P. Beaumier. Simulation of unsteady aeroelastic response of a multibladed rotor in forward flight. In *proceedings of the* 27^{th} *European Rotorcraft Forum, Moscow*, 11-14 september 2001.

[7] H. Ding, C. Shu, K.S. Yeo, and D. Xu. Development of least-square-based two dimensional finite-difference schemes ans their application to simulate natural convection in a cavity. *Computers and Fluids*, (33):137–154, 2004.

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

[9] G. Jeanfaivre, C. Benoit, and M. C. Le Pape. Improvement of the Robustness of the Chimera Method. In *proceedings of the* 32^{nd} *AIAA Fluid Conference and Exhibit*, Saint Louis, Missouri, USA, 2002.

[10] G. Jeanfaivre, X. Juvigny, and C. Benoit. Parallel chimera computations of helicopter flows. In *proceedings of the* 24^{th} *Congress of the International Concil of the Aeronautical Sciences (ICAS)*, Yokohama, Japan, 2004.

[11] X. Juvigny, E. Canonne, and C. Benoit. Multigrid algorithms for the chimera method. In *proceedings of the* 42^{nd} *AIAA Aerospace Sciences Meeting*, Reno, Nevada, USA, january 2004.

[12] A. Lerat, J. Sidès, and V. Daru. An implicit finite-volume method for solving the euler equations. In *Lecture Notes in Physics*, volume 170, pages 343–349, 1982.

[13] R.L. Meakin. Unsteady Aerodynamic Simulation of Mutiple Bodies in Relative Motion. AIAA 89-1996, 1989.

[14] R.L. Meakin. Adaptive Spatial Partitioning and Refinement for Overset Structured Grids. *Computer Methods in Applied Mechanics and Engineering*, 1999.

[15] S. Peron, C. Benoit, and G. Jeanfaivre. The constraint-free chimera using a gridless approach. In *International Conference on Adaptive Modelling and Simulation, ADMOS 2005, Barcelona*, September 2005.

[16] N. C. Prewitt, M. Belk, and W. Shyy. Improvements in parallel chimera grid assembly. *AIAA Journal*, 40(3):497–500, March 2002.

[17] T. Renaud, C. Benoit, J.C. Boniface, and P. Garderein. Navier-stokes computations of a complete helicopter configuration accounting for main and tail rotors effects. In *proceedings of the* 29^{th} *European Rotorcraft Forum, Friedrichshafen*, 16-18 september 2003.

[18] T. Renaud, D. O'Brien, M. Smith, and M. Postdam. Evaluation of isolated fuselage and rotor-fuselage interaction using cfd. In *proceedings of the 60th AHS Annual Forum*, 7-10 june 2004.

[19] B. Rodriguez, C. Benoit, and P. Gardarein. Unsteady computations of the flowfield around a helicopter rotor with model support. In *proceedings of the* 43^{rd} *AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada*, January 2005.

[20] T. Schwarz. Development of a Wall Treatment for Navier-Stokes Computations using the Overset Grid Technique. In *proceedings of the* 26^{th} *European Rotorcraft Forum, The Hague*, September 2000.

[21] S. Yoon and A. Jameson. An LU implicit scheme for high speed inlet analysis. In *AIAA Paper 86-1520*.