TRANSONIC FLOWS, SHOCK WAVE — TURBULENT BOUNDARY LAYER INTERACTION

PIOTR DOERFFER AND JAROSŁAW KACZYŃSKI

Institute of Fluid Flow Machinery
Polish Academy of Sciences
Fiszera 14, 80-952 Gdansk, Poland

Abstract: Shock wave - boundary layer interaction is one of the most important phenomenon in transonic flows. Due to its complexity it is difficult as well for experimental as for numerical study. The growing potential of the CFD is therefore of high importance. Different aspects of shock wave - boundary layer interaction should be studied in different flow configurations. Therefore results concerning profile flow, helicopter rotor at hovering and forward flight and internal flows are presented in this paper. These are to illustrate our ability in CFD in general. Besides the flows simulation the work directed to a development of used codes is carried out.

1. Introduction

The Transonic Flows and Numerical Methods Group of IMP PAN (TFNM) was established at the beginning of 1996 and at present is one of the most active users of TASK resources in the field of CFD.

Long lasting period of poor or lacking access to supercomputers caused an absence of Navier-Stokes codes development in Poland in recent decades. Growing potential of workstations and mainframe computers in the Western World caused that the Navier-Stokes solvers were adopted for these computers. It became cheaper than supercomputer application.

Fast development of computer resources in Poland through the establishment of centres like T.A.S.K. has provided enough computational power to treat flow problems by means of Navier- Stokes solvers. It has, however, appeared that there are no codes and those available on the market cost a lot of money and being in a closed form leave very little space for personal scientific input into its development.

Having well established co-operation with two research centres:

- DLR (German Aerospace Establishment)
- University of Karlsruhe

in Germany it has been agreed to incorporate the TFNM into the work on numerical simulations and further development of Navier-Stokes solvers used in these centres. The sources of the codes have been transferred and so all aspects of research on CFD became available. Frequent meetings with authors of the codes provide a very important support. From DLR the ViB code has been allowed for our use and from Karlsruhe the KAPPA code is accessed.

The research directions in the Group are derived and closely linked to the experimental work. Newly built Transonic Wind Tunnel provides exceptional possibilities in this respect. The times when some researchers considered CFD to be able to substitute experiment are far behind us. The more advanced CFD methods are, the greater the need for good experiment is. TFNM is making a lot of effort to combine CFD and experiment.

The undertaken research directions in CFD are threefold:

- numerical simulations of chosen flow cases,
- adoption of solvers for some flow configurations,
- implementation of new turbulence models.

The results presented in this paper have been obtained using KAPPA solver which has been fully described by its author Dr Magagnato in a separate paper in this issue. Therefore no description of the numerical methods has been placed in the paper presented here.

2. Numerical Simulation Environment

Pre-processing:

Integrated Grid Generation system from NUMECA, originating from Prof. Hirsch group from Belgium (VU Brussels) has been purchased. It provides an interactive graphical environment for:

- surface modelling,
- grid generation.

It allows a generation of 2D and 3D multi-block grids for structured meshes and 2D grids for unstructured ones [1].

Post-processing:

Due to the use of SGI computers at the beginning the VISUAL3 from public domain has been utilised. It has been created at MIT by Bob Heims as the main author of the system. It is mainly dedicated to the visualisation of numerical results of fluid flow simulation. The system is very useful for 3-D structured and unstructured grids and for stationary and time dependent flows [2].

On transferring of calculations to T.A.S.K. computers the possibility to use AVS — EXPRESS (Advanced Visual Systems) emerged. It has been adopted for some of our flow cases visualisation.

Solvers:

There are three solvers used:

1) ViB-IMP (Viscose Block-structured Code) originally from DLR Göttingen. It is used in cooperation with Dr Schwamborn.

Block structure, 3-D and 2-D with multigrid, compressible and steady flows.

2) KAPPA (Karlsruhe Parallel Program for Aerodynamics) ISL, Universität Karlsruhe in cooperation with Dr Magagnato.

Block structure, 3-D multigrid and multilevel, compressible, steady and unsteady flows, parallel multiprocessor calculations.

ViB and KAPPA codes integrate the compressible, Reynolds-averaged Navier-Stokes equations using a cell-centred finite-volume and explicit Runge-Kutta time stepping method.

ViB provides only the Baldwin-Lomax turbulence model and now a new model of Spalart-Allmaras has just been implemented into the code.

KAPPA is equipped with algebraic, as well as two equation models. It implies full multigrid and provides several numerical schemes. It allows for parallel calculations on multiprocessor machines. This important feature uses Message Passing Interface (MPI) which has been installed in TASK.

3) KACZYNSKI solver developed in IMP PAN presented in doctor thesis [3]. It is for stationary, 2-D flows of incompressible fluid. It allows simulations of laminar flows or turbulent with k-e turbulence model and is based on control volume method for non-orthogonal and non-staggered meshes.

This solver has been validated in the ERCOFTAC Workshop on Data Bases and Testing of Calculation Methods for Turbulent Flows; April 3-7, 1995, Karlsruhe, organised by Prof. Rhodi from Karlsruhe. Results obtained without any previous knowledge of experimental data were one of the best among research centres of very good reputation.

3. Research Directions

Transonic Flows and Numerical Methods Group of IMP PAN deals with problems of flow in which at least a part is supersonic. The most characteristic feature of supersonic flows is that compression always takes place as so called „shock wave". It means an abrupt change of flow parameters. Such sudden change of flow parameters introduces a very nasty challenge to numerical methods.

The existence of shock waves also introduces an interesting physical phenomenon which is its interaction with boundary layers, always present at walls limiting any flow. The interaction is important because it takes place in many modern designs. The most familiar example is the airfoil of commercial planes. On the upper side of it a local supersonic area is formed which is terminated with a normal shock wave (Figure 1). In the case of a helicopter rotor at regular flight velocity a supersonic flow is also induced at the advancing blade (Figure 2.) [4].

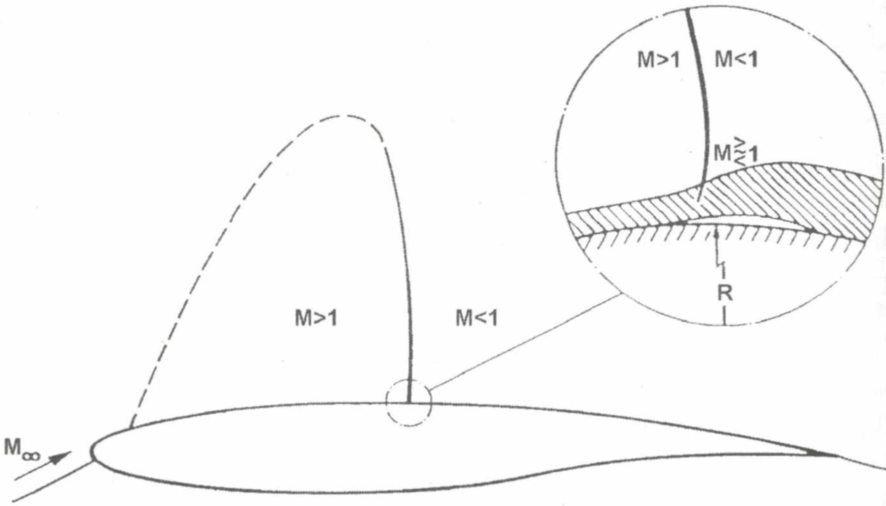


Figure 1. *Typical transonic profile with local supersonic area and terminating shock wave*

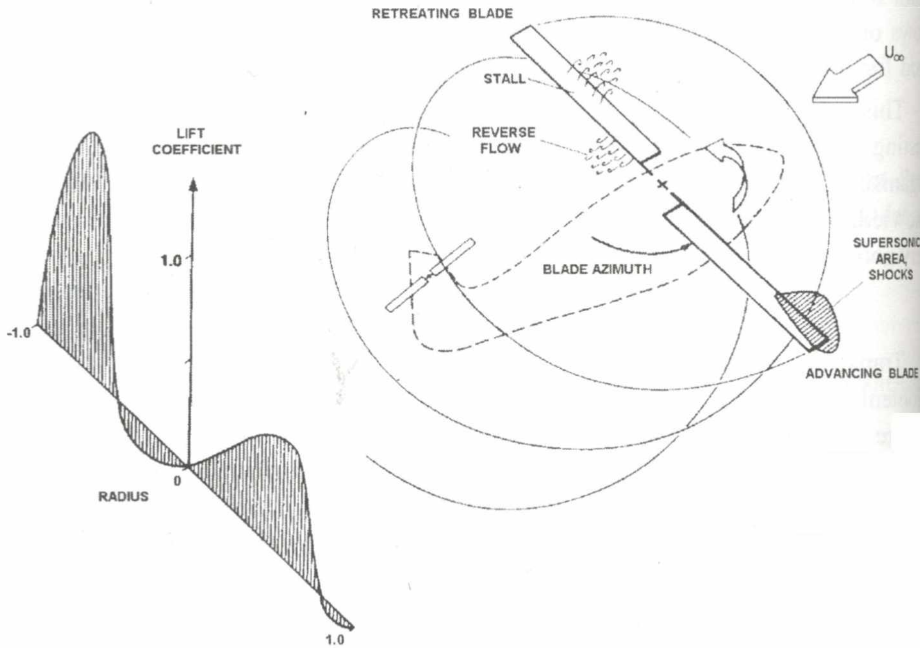


Figure 2. *Supersonic area on helicopter rotor* [*4*]

It is a source of significant noise and therefore draws the attention of researchers.

Another area of interest are internal flows. The shock wave stretches then across the channel and introduces much stronger consequences than in external flows. Such flow problems appear in supersonic intakes to air breathing engines and in blade passages of gas and steam turbines. It also takes place at an outlet of any standard pressure system where air is compressed to more than 2 bar. A typical example of such flow problem is a shock wave in a convergent-divergent channel (Figure 3), so called Laval nozzle. A characteristic feature of transonic flow is that

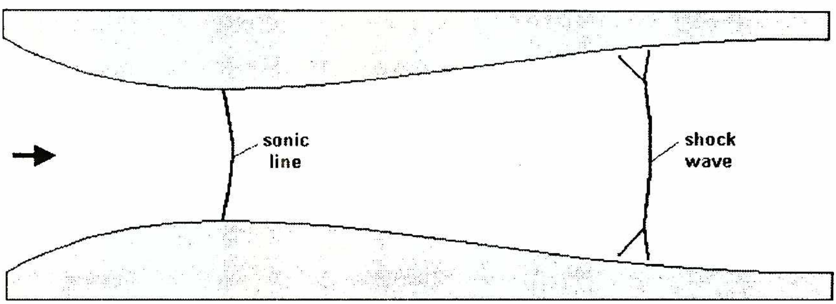


Figure 3. *Laval nozzle, flow topography*

in the place of smallest cross section area (throat) a sonic velocity is reached and further downstream a supersonic expansion is continued. At certain outlet pressures a shock wave is generated in the passage. The shock wave interacts with a boundary layer on the walls. One of the noticeable effects of this phenomenon is shock splitting in the form of a λ-foot for $M>1.4$.

In all technical applications mentioned here boundary layers are turbulent. The strongest type of shock is a normal one. Therefore, normal shock wave-turbulent boundary layer interaction is in the centre of interest. The importance of the interaction phenomena is emphasised by its presence at the nominal working conditions of the units mentioned above.

Shock wave - boundary layer interaction could be investigated under many aspects [5-10]. At present there are three main directions of the research carried out:

- shock configuration; λ-foot formation,
- shock induced incipient separation,
- development of separation and flow structure of separated flow.

Numerical simulations became a very important tool for the research in the field of science mentioned above. The work on CFD is rather in the preliminary stage in our Group. The results obtained, however, are interesting and worth presenting because they show the ability of the KAPPA to solve problems of our interest.

4. Transonic Flow around a Profile.

One of the very essential problems in the numerical simulations of viscose flow is the grid generation. It has to allow sufficient resolution in the boundary layer and also in the area of a shock wave. The quality of grid significantly affects the convergence speed. General grid layout is presented in Figure 4. Grid type „C" is the best for profile flow simulation. Flow direction is from left to right (it is a rule for the illustrations presented here).

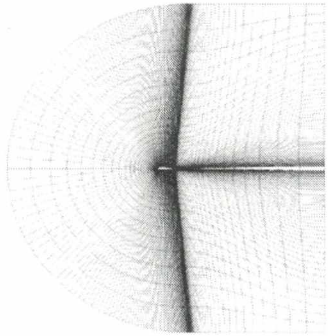


Figure 4. *General grid view for profile simulations*

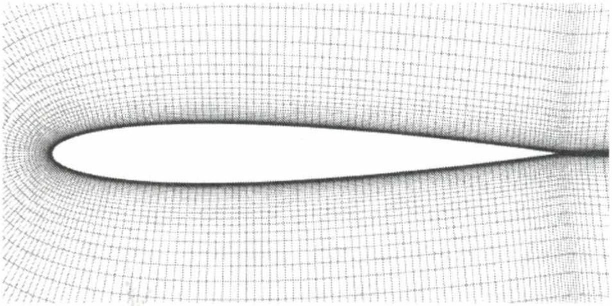


Figure 5. *Classical NACA 0012 profile*

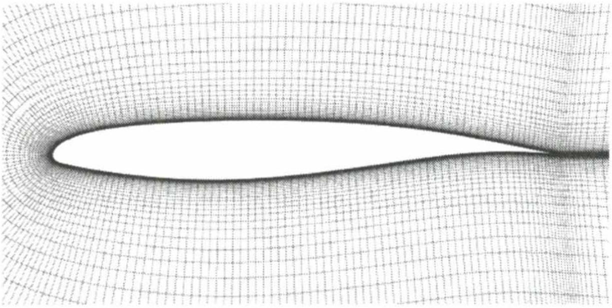


Figure 6. *Transonic profile CAST 7*

For introductory calculations a standard profile NACA 0012 (Figure 5) and also a typical transonic profile CAST 7 have been used (Figure 6). The grids were prepared so that the point closest to the profile is in a distance of about y+ = 1. Such resolution is important in order to obtain all the details concerning boundary layer, hence to obtain good results concerning pressures and forces.

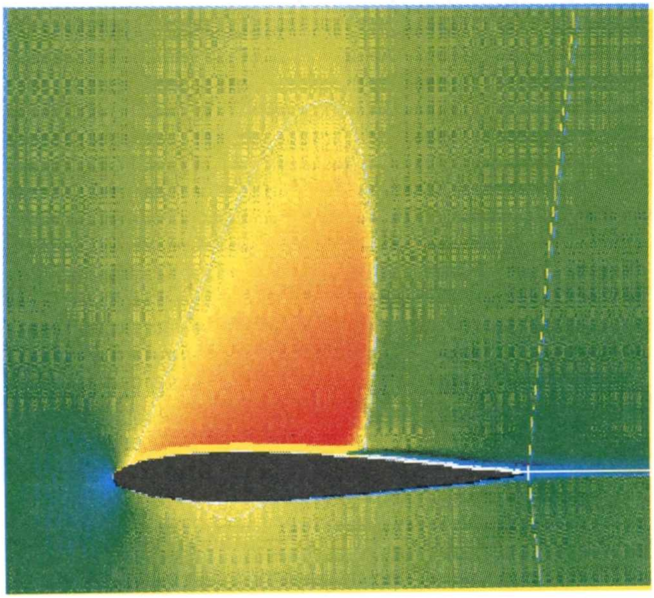


Figure 7. Transonic profile flow simulation

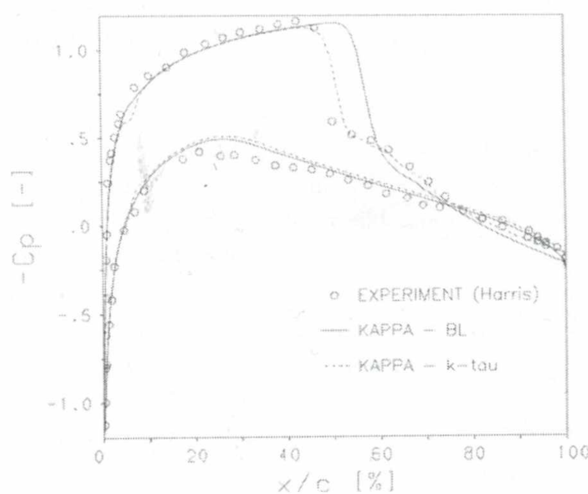


Figure 8. Pressure coefficient distribution on the NACA0012 profile at α = 2.26° and M = 0.799

A typical flow field structure is presented in Figure 7 as a contour plot of iso-Mach number lines. A white line indicates the sonic line (M=1.0), except for two straight lines originating from the trailing edge which are grid block borders. As it has been mentioned above, supersonic area is terminated by a shock wave which interacts with the boundary layer on the profile surface. The quality of obtained results is dependent on the flow complexity. In subsonic flow it is simulated very well but in the case of transonic flow conditions, when shock wave appears, the numerical results coincidence with experiment is not as good. It also becomes sensitive to turbulence models. In Figure 8 a comparison of pressure coefficient distribution on the profile is presented. It concerns experiment and numerical simulations with two turbulence models. Boldwin-Lomax algebraic model indicates significant downstream shift of the shock wave location in respect to the experiment. In the case of k-τ two equation model shock wave location coincides much better with experiment.

In the research carried out the profile flow is mainly used for comparison of simulations with newly implemented turbulence models. This is because this flow configuration is very well documented and a lot of numerical and experimental material is available for comparison.

5. Helicopter Rotor Dynamics

One of the cases of transonic flow appearance is the advancing blade of helicopter rotor. It becomes one of the important noise sources which draws a lot of attention. There are certain ideas of approach to reduce the shock wave-boundary layer interaction intensity. At present an effect of passive control of the interaction on an airfoil is under investigation [11]. It is of course treated as a local influence. In the case of a helicopter rotor the lift at an advancing blade is too high and therefore it would be acceptable to disturb the whole supersonic area. Such an approach may bring some improvement, we hope.

In order to be able to investigate a flow around helicopter rotor a possibility of adding grid velocity has been used at every grid point. At test calculation rotor tip section Mach number was M = 0.7 and flight M = 0.2. These are very hard conditions because the advancing blade tip section speed is M = 0.9. The unsteady calculation carried out is using dual time stepping. For this preliminary calculation each grid consists of about 1.2 million grid point and is splitted into 30 blocks. Rotor geometry is very simple, based on ellipse at the root and a straight line at the tip, with 7.5 blade aspect ratio. This simple geometry, with extremely thin profile at the tip, has allowed to avoid problems of complicated flow structure calculations and therefore has provided relatively simple control of the simulations' correctness.

Figure 9 and Figure 10 illustrate obtained results on a rotor in the position normal to the flow direction, shown in perspective view. Static pressure distribution on the rotor surface is shown by a colour. High pressure in red denotes leading edge. Between two lines on the rotor a middle part is located with the centre of rotation.

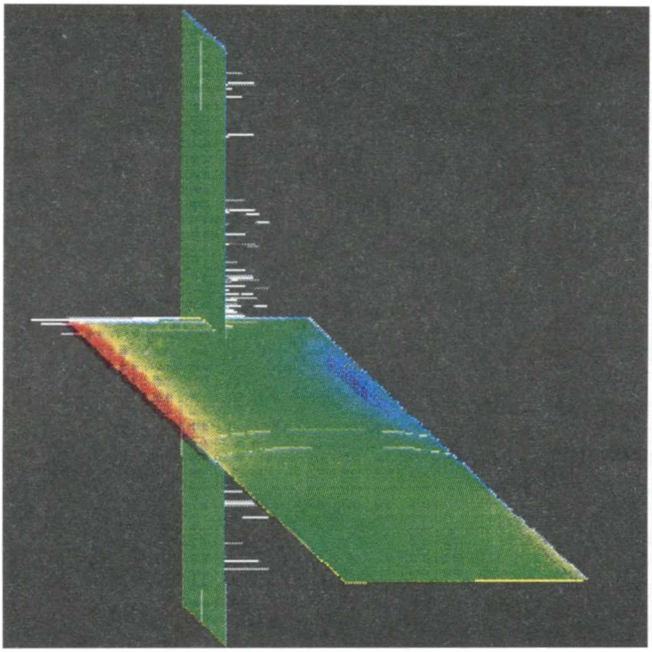


Figure 9. *Helicopter rotor with a cutting plane at the advancing tip*

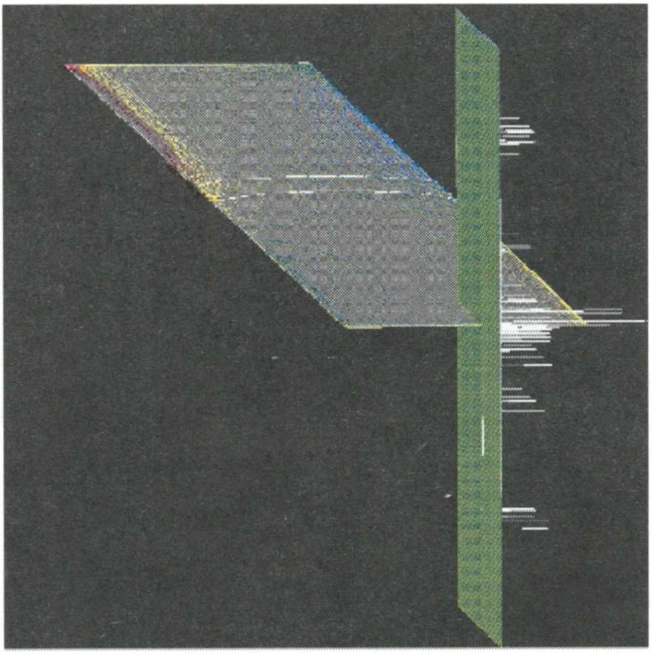


Figure 10. *Helicopter rotor with a cutting plane at the retreating tip*

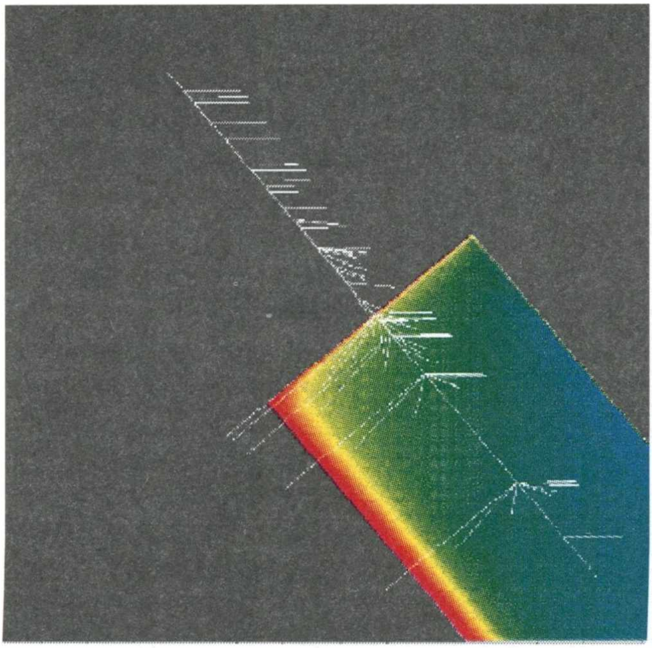


Figure 11. Helicopter rotor with a cutting plane at the advancing tip, 36°

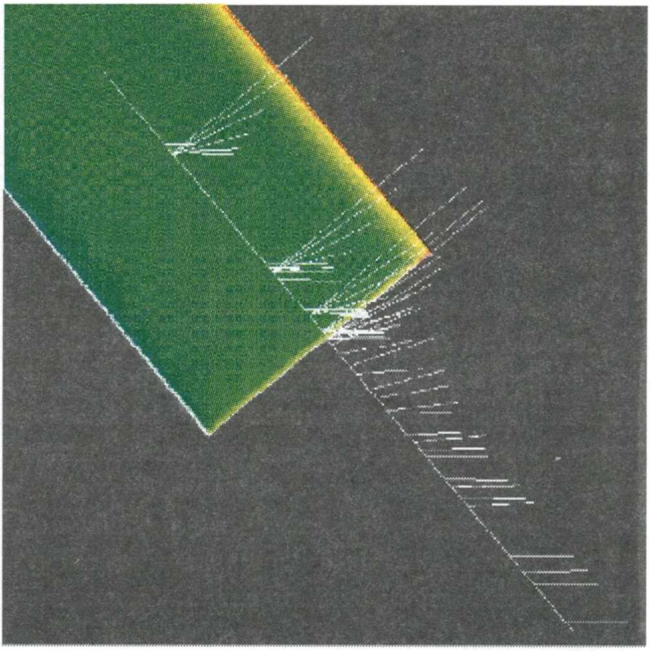


Figure 12. Helicopter rotor with a cutting plane at the retreating tip, 36°

It may be observed that advancing (further) blade achieves higher pressures than the retreating blade (closer). The leading edge transfer point from one blade side to the other is located on the retreating blade part. This displacement of the zero velocity blade section is caused by the rotor advancing velocity. In Figure 9 the advancing tip is cut by a spanwise vertical plane. On this cutting plane velocity vectors are plotted as white lines in 3-D. It should be emphasised that the motion is related to the rotation axis. That means that away from the rotor, in so called far field, Mach number is equal to M = 0.2 and at the blade due to non-slip condition in viscous flow velocity is equal to rotor velocity (depending on radius) equal to M = 0.7 at the tip itself. At the advancing tip free stream velocity vectors have a direction opposite to the rotor blade. In Figure 10, for retreating blade tip, the free stream and blade velocity vectors are in the same direction. Therefore the relative blade velocity is smaller and stagnation pressure reached at the leading edge is smaller. It is well displayed by the colour of pressure.

The situation looks more complex in different rotor positions. In Figure 11 and 12 the rotor is inclined by 36° from the cross-flow position. In these figures a view from above is presented on the tip end of the advancing (Figure 11) and retreating (Figure 12) blades, in which a vertical cutting plane is seen as a spanwise line. In the case of presented blade position the velocity vector changes its direction significantly within the boundary layer. The flow is strongly three dimensional. In Figure 11 the pressure colour indicates that at this rotor location the tip of blade contour becomes a part of leading edge too.

Obtained results indicate that the KAPPA is able to simulate successfully the hovering and flight of a helicopter rotor. Further work planned for the nearest future is the implementation of a chosen blade profile and simulations at hovering condition. Later when the flight is taken into account a variable angle of attack of blades, depending on its angular position will have to be taken into account.

6. Internal Flows

Shock wave-boundary layer interaction is also very important for internal flows. It gains even on importance as all experimental work is being done in wind tunnels which belong to internal flows.

In Figure 13 an experimental visualisation of a shock wave in a Laval nozzle has been shown at M=1.47. Flow direction is from the left to the right side. Upper and bottom walls are divergent and outlet pressure is set up so that a shock wave is in view. It stretches across the channel. The existence of boundary layers at walls (seen as a white strip at the lower wall) is responsible for splitting of the shock close to the wall, and formation of a so called λ-foot. The presented flow is obtained in a passage where the change of its shape is formed by the mentioned upper and bottom walls. Side walls are parallel and are equipped with glass windows, allowing for presented visualisation. In such configuration and visualisation one easily gets an impression that the flow is 2-D. In reality mentioned here λ-foot exists also near side walls. Due to this the shock induced separation takes place at all four walls of the channel. It is shown in Figure 14 by means of the oil visualisation technique at tunnel walls. The separated flow is very complex and obviously influences the flow in the whole channel. This must be also the reason for discrepancies between 2-D calculations of the flow and the experiment in case of appearance of separation.

As it has been mentioned, a shock wave-boundary layer interaction is in the centre of interest under many aspects. Planned numerical simulations will need concentration on numerical schemes and turbulence modelling, appropriate for the investigated topic. At present following features of nozzle flow have been tested:

- 2D nozzle; structure of nozzle flow; shock wave location depending on downstream pressure condition
- 3D nozzle; ability of separation flow structure resolution

In Figure 15 a typical 2-D divergent-convergent flow structure obtained in numerical simulation is presented. Flow is from left to right and the colours correspond to Mach number values. Downstream the throat a sonic line (white)

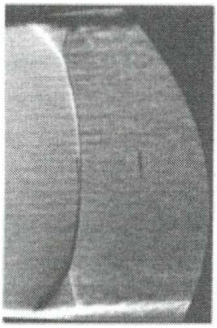


Figure 13. *Schlieren visualisation of shock wave in a Laval nozzle.*

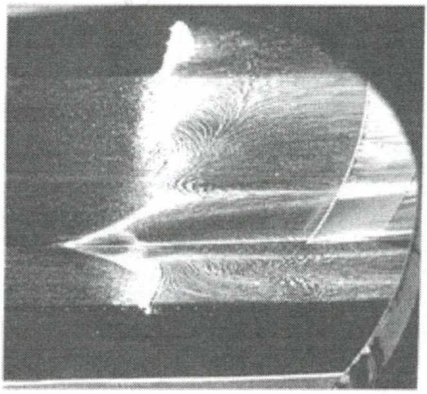


Figure 14. *Oil flow visualisation of separation flow structure corresponding to Figure 13*

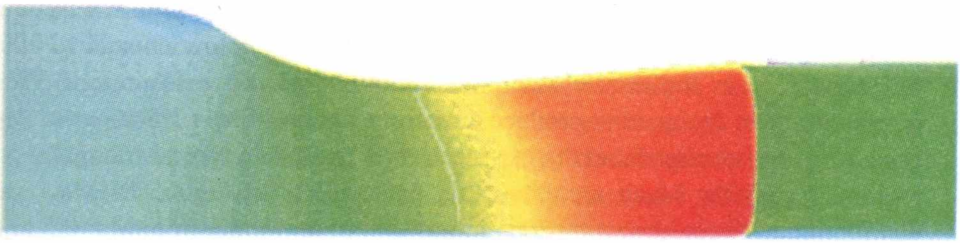
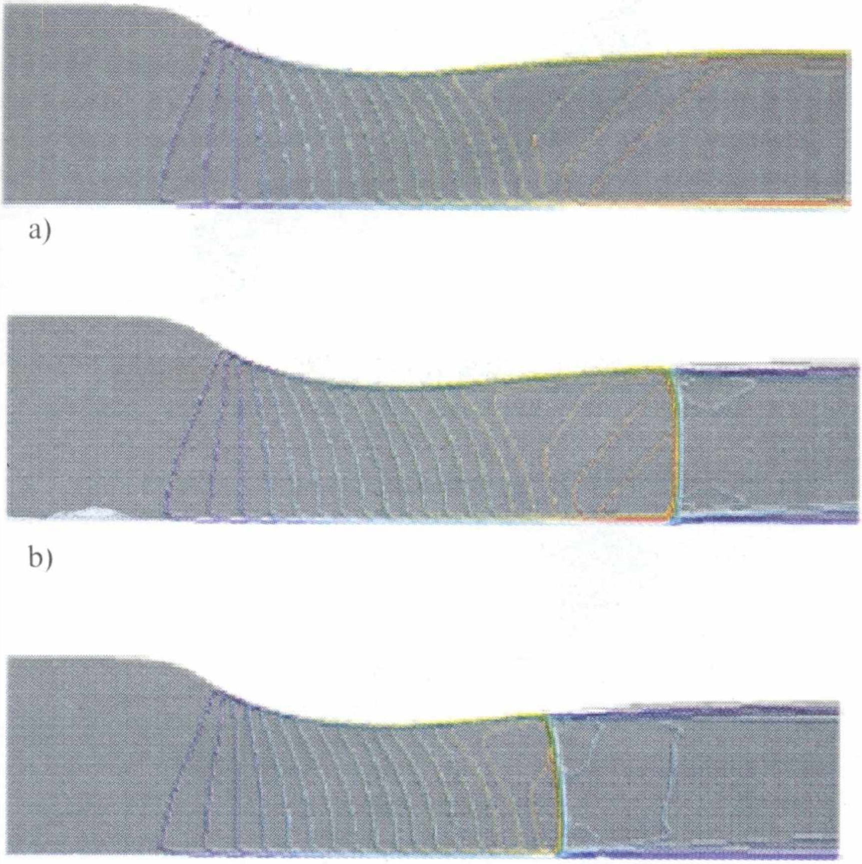


Figure 15. *An example of 2-D simulation of nozzle flow with a shock wave*



a)

b)

c)

Figure 16. *2-D nozzle flow for different outlet pressure: a) low outlet pressure - full expansion, b) increased outlet pressure, shock upstream Mach number M = 1.47, c) further increased outlet pressure, shock upstream Mach number M = 1.35*

shows a typical curvature. Further downstream an abrupt change of parameters in the form of a shock wave appears. Close to the walls boundary layers are present and shock wave indicates an upstream influence. A fine resolution of λ-foot structure will demand some more attention, however all the qualitative features including post expansion are displayed.

As Figure 16 indicates the downstream pressure condition has a qualitatively correct effect. In Figure 16a the downstream pressure is low enough to allow for full expansion in the nozzle and therefore no shock wave is present. Presented nozzle has been designed to produce a uniform stream of M = 1.5 at outlet. This has been very well obtained and the iso-Mach lines indicate a typical triangular outflow region of constant Mach number value. In all plots of Figure 16 Mach number distribution has been shown by 20 contours spaced uniformly from M = 0.5 to 1.5.

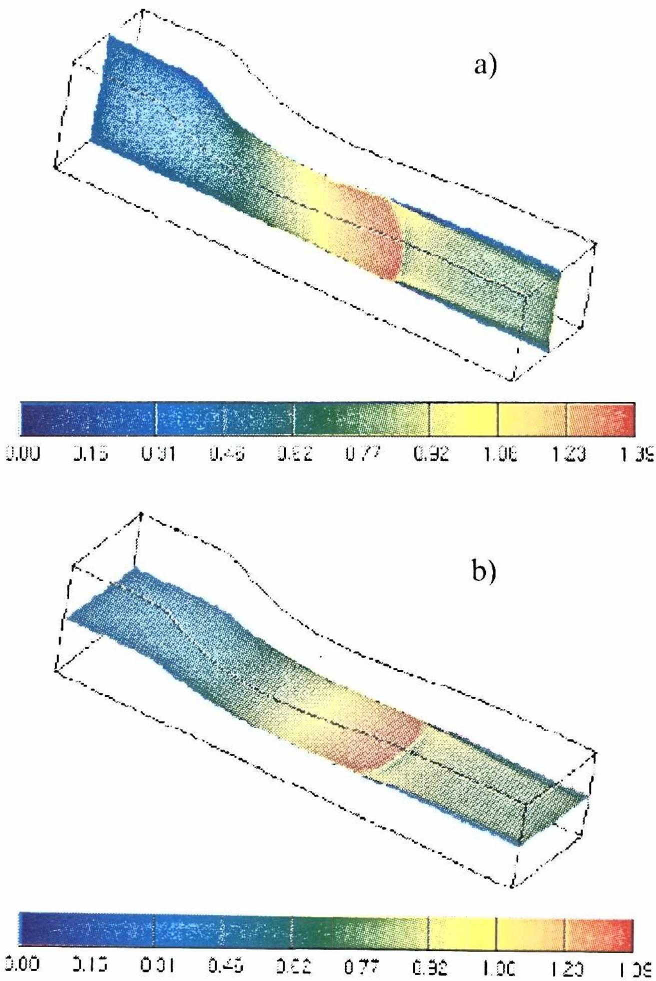


Figure 17. *3-D nozzle flow a) vertical middle plane iso-Mach contours visualisation, b) horizontal middle plane iso-Mach contours visualisation*

Figs16b and c show the cases of increasing outlet pressures. When pressure does not allow for full expansion but is low enough to induce supersonic flow downstream the throat a shock wave must appear. This shock wave is shifting upstream with increasing outlet pressure. Simultaneously shock upstream Mach number decreases. This is well documented by numerical simulation.

In order to analyse the structures of shock induced separation a 3-D approach is necessary. It introduces severe difficulties through the mesh size. If one wants to resolve separation a very fine resolution of mesh near the wall is necessary and, as our experience shows, one easily ends up with a million mesh points.

In Figure 17 vertical (17a) and horizontal (17b) middle planes are presented showing iso-Mach number colours. One can easily recognise all the qualitative details of shock wave-boundary layer interaction as in 2-D simulations.

The 3-D separation structures are detectable, which is shown in Figure 18. The presented visualisation needs some explanation. It is done in such a way that very close to a nozzle wall a plane is located on which a certain mesh of points is chosen. For a selected time interval flow elements passing through each mesh point on the plane are marked. By means of this procedure one obtains parts of 3-D streamlines originating from the points on the plane. Their length depends on the velocity. Figure 18a shows a visualisation at a side wall. The plane starts shortly upstream the separation area. The used mesh of marking points is easy to recognise. Empty spaces are zones where flow velocity is very low. It is observed that some streamlines are interlaced into the separation area and are then leaving the passage. This is a typical behaviour in 3-D. Asymmetry of the separated flow is due to the channel asymmetry. Figure 18b shows the same type of visualisation but at the

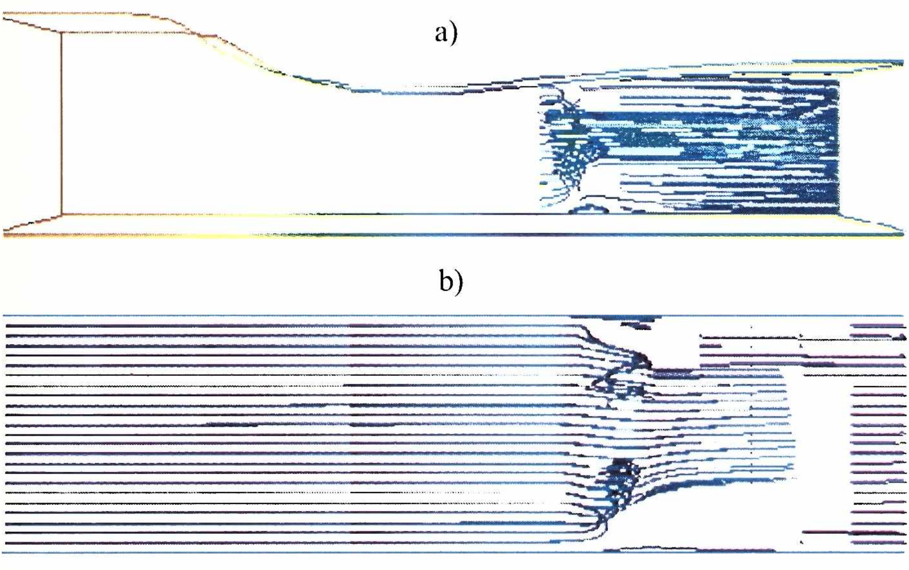


Figure 18. *Streamlines visualisation in wall vicinity, a) Side wall, b) Bottom wall*

bottom wall. This time the marker plane stretches from the channel inlet. A very interesting observation is the asymmetry at this wall. It has to be investigated in more detail. In experiments asymmetry at this plane may be also observed but we have considered it to be en effect of some inaccuracy of a channel preparation. Therefore, it has not been expected in numerical simulations. The result obtained may mean, however, that there is another physical reason for this asymmetry.

7. Conclusions

The results presented above indicate that the potential of numerical simulation of flow problems undertaken in the Research Group is on the levels of present world-wide standards. The success of research on selected topics depends solely on the ability to use and develop existing solvers.

Additional research direction which has not been undertaken yet is unsteady flow. It is especially important for transonic flow conditions. We are, however, not able to undertake this very popular research direction due to the lack of manpower at present.

References

[1] K. Namieśnik, *Generacja siatek – prezentacja programu IGG*, nr.arch 451/97

[2] A. Prońska, *Interfejs KAPPA – Visual 3*, nr arch.: 466/97

[3] J. Kaczyński, *Numerical analysis of dissipative phenomenon in selected flow configurations*, doctor thesis, IMPPAN, October 1995

[4] F.X. Caradona *Application of Transonic Flow Analysis to Helicopter Rotor Problems*, Progress in Astronautics and Aeronautics, Vol. 120, pp: 263-285

[5] P. Doerffer, J Zierep *An experimental investigation of the Reynolds number effect on a normal shock wave – turbulent boundary layer interactions on a curved wall*, Acta Mechanica 73, pp. 77-93 (1988)

[6] P. Doerffer, *An experimental investigation of the Mach number effect upon a normal shock wave – turbulent boundary layer interaction on a curved wall*, Acta Mechanica 76, pp. 35-51 (1989)

[7] P. Doerffer, *Normal shock *-foot topography at turbulent boundary layer*, Acta Mechanica (1994) [Suppl] 4: pp133-140

[8] P. Doerffer, J. Amecke, *Secondary flow control and streamwise vortices formation*, ASME TURBO-EXPO '94, International Gas Turbine and Aeroengine Congress and Exposition, the Hague, Netherlands - June 13-16, 1994, ASME Paper 94-GT-376

[9] P. Doerffer, U. Dallmann, *Reynolds number effect on separation structures in normal shock wave-turbulent boundary layer interaction*, AIAA Journal, Vol. 27, No. 9, 1989

[10] P. Doerffer, U. Dallmannn, *Secondary corner and channel flow effects on shock induced turbulent flow separation*, Proceedings of the IUTAM Symposium on Separated Flows and Jets, Novosybirsk, 1990, Springer Verlag

[11] P. Doerffer, R. Bohning, *Passive control of a normal shock wave-turbulent boundary layer interaction*, RRDPAE '96, Warsaw 1996, 25-27 November