NEW JET-ENGINE-INLET BOUNDARY CONDITION FOR THE COMPUTATION OF NAVIER-STOKES FLOWS AROUND NACELLES OF JET ENGINES

This final-thesis project concerned the computation of compressible flows around a modern isolated nacelle of a jet engine for a transport aircraft. The flow fields included flows in the nacelle inlet upstream of the engine inlet surface, and downstream of the engine outlet surfaces. The turbulent boundary layers in the nacelle inlet and over the exterior of the nacelle surface were computed. The engine nacelle was that for a modern turbine-powered simulator of an aircraft jet engine.

The Navier-Stokes flow solver ENSOLV of the National Aerospace Laboratory NLR in the Netherlands was used to compute the flows. For this project, the flow solver had to be executed on a large parallelel vector supercomputer (the NEC SX4 of NLR), to obtain sufficient computation power, because very fine three-dimensional grids were required to obtain computation results with acceptable accuracy.

The primary goal of the final-thesis project was the development of a stable numerical solution algorithm, in the Navier-Stokes flow solver ENSOLV, for a boundary condition at the jet-engine-inlet surface. The mass flux through this surface into the jet engine should be allowed to have prescribed values. A complication was the fact that the boundary layers in the nacelle inlet surfaces entering the engine through this surface should also be taken into account.

The flow solver ENSOLV is part of a complete flow-simulation system of the National Aerospace Laboratory NLR. This flow-simulation system is called ENFLOW. Fig. 1 shows an overview of the ENFLOW system. The system consists of three main parts:
ENDOMO: A domain modeller. The input of ENDOMO is the geometric shape of a 3-dimensional flow dmain. ENDOMO may be used to subdivide the flow domain around the, often complex, geometry into simple subdomains, called blocks. Each block has the shape of a deformed cube.
ENGRID: ENGRID may be used to construct within each block a structured grid, so that a multi-block grid in the comlete flow domain is obtained. The grid density must be chosen high in regions with large flow gradients, to get computation results with a reasonable numerical accuracy.
ENSOLV: With a given multi-block grid, the flow solver ENSOLV computes the solution of 3D Navier-Stokes flow equations and the specified boundary conditions.
For the visualization of the flow results, there are a number of software products, like VISU3D, FLOVIS, PINDA, Data Visualizer, gnuplot, and PV-Wave.

A time-dependent boundary-condition equation was designed and tested numerically. When the flow computation converges to a steady-state flow, this boundary condition should allow the mass flux through the inlet surface in steady flow to have prescribed values. A time-dependent boundary condition could be designed, because the iteration algorithms in the flow solver ENSOLV are based on local time stepping with Runge-Kutta like algorithms, which are accellerated by multigrid by and smoothing by implicit-residual averaging.

To thoroughly test the new numerical solution algorithm for the engine-inlet boundary condition with flow computations, a fine multi-block grid around the jet engine was produced, with a total of 2.17 million grid cells. This grid was designed such that nonlinear instabilities with the flow-computation algorithms towards steady flow states, which were due to the grid, did not appear. Using this grid, it could subsequently be concluded that the new solution algorithm was stable (flow-equation residuals for steady flows could be made as small as desired).

The convergence of steady-flow computation results with mesh size was also satisfactory. Repeated mesh-size halving in all three space directions was applied to obtain rigorous conclusions about the effects of grid-size reductions. Very fine grids are necessary to compute, with at least some numerical accuracy, the complex flow phenomona (shocks and various kinds of shear layers) that may turn up in exhaust jets.

Results of wind-tunnel tests were used to validate aerodynamically the new solution algorithm for the inlet boundary condition.

Figure 2 gives a cross-sectional view of the nacelle around the jet engine. In this project, the engine and the nacelle had axial symmetry. However, the final grid was not made axi-symmetric, because this was found to lead to non-linear instabilities of the flow computations towards steady-state flows. The flow computations were done in three dimensions, not only because the grid was not axi-symmetric, but also because, in a number of flow computations, the nacelle and engine were placed under incidence with respect to the oncoming free streams, so that the steady flow was also not axi-symmetric.

Figures 3 through 4 show the pressure-coefficient and Mach-number distributions in the inner and outer exhaust jets of a computed subsonic-transonic flow. The flow is in this example axi-symmetric. There are shock waves, velocity-jump and temperature-jump shear layers, and there is, at the outlet plane of the outer jet, acceleration of a subsonic flow through sonic conditions to supersonic flow. The flow around the nacelle and exhaust jets is subsonic. Figures 5 through 6 show details of the multiblock grid for this flow computation. The fine grid is the grid with the 2.17 million grid cells. The medium grid was derived from the fine grid by omitting every other grid line, in each of the three coordinate directions.

The results of the study were:

- A new, stable solution algorithm for a time-dependent jet-engine-inlet boundary condition for the Navier-Stokes equations.
- A special multi-block grid in a computational domain around the jet engine, where grid singularities are avoided.
- Favourable conclusions on grid convergence.
- Favourable conclusions on convergence towards steady flows.

- Computation results which compare well with corresponding data of wind-tunnel tests.
- Insight in grid resolutions, which are required to capture, at least with some numerical accuracy, vortex formation around exhaust jets, when these are inclined with respect to the surrounding flow, and complex flow phenomena that can arise in exhaust jets.

The new boundary condition opens new possibilities to use CFD for the computational aerodynamic analysis of flows around engines. The boundary layers, formed on inlet walls, affect the performance of jet engines. A proper accounting of the effect of the boundary layers in the inlets is therefore of great interest for engine nacelle and inlet conditions (engine-nacelle angle of attack about $0°$, high Mach-number flows), but also during take off and in climb (angles of attack of the order of $10°$ and more, at lower flight speeds). There is a risk of boundary-layer separation on the inside of the lower lip of nacelle inlets, due to suction peaks and even shock waves. Boundary-layer separations in nacelle inlets have unfavourable effects on the economy and operational stability of engines.

The results of the final-thesis stage study have been reported in the NLR Technical Report TR 97007 L: "Design of an Engine Inlet Boundary Condition for the Navier-Stokes equations for the Numerical Simulation of Jet Engines", by T. Esman.

For questions and remarks, please contact C. Vuik, Department of Applied Analysis, Faculty of Technical Mathematics and Informatics, Delft Universtity of Technology, Delft, the Netherlands, email c.vuik@math.tudelft.nl, or J.W. Boerstoel, Theoretical Aerodynamics Department, National Aerospace Laboratory NLR, Amsterdam, the Netherlands, email boerstl@nlr.nl.

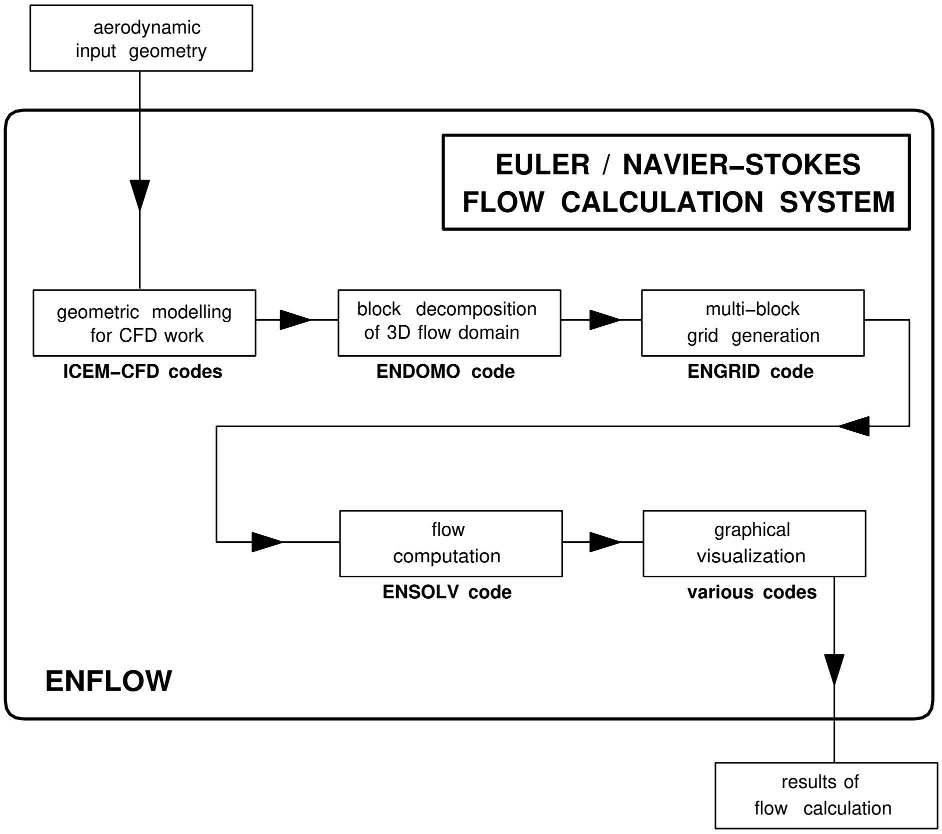

Figure 1: The Navier-Stokes flow-computation system ENFLOW of NLR, with the ENSOLV flow solver.

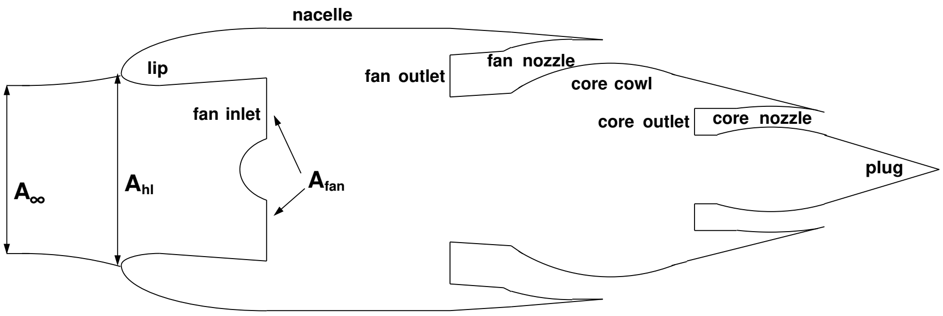

Figure 2: Engine and nacelle parts of typical turbofan jet engine.

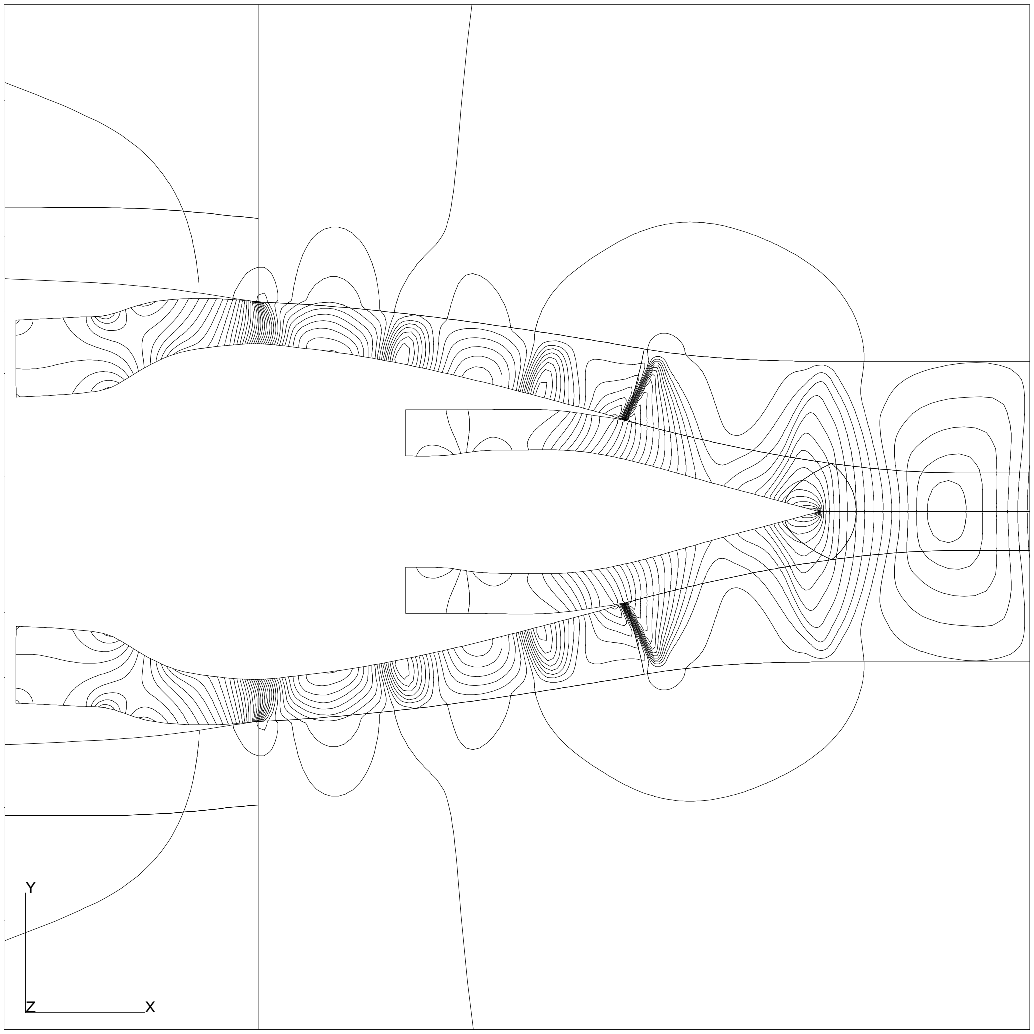

Figure 3: Lines of constant pressure coefficient in the vertical symmetry plane in outlet area. Axi-symmetric flow. Fine grid. Contour lines in pressure coefficient 0.1 apart. The three regions of sharp pressure rise are shock waves in the outer exhaust jet.

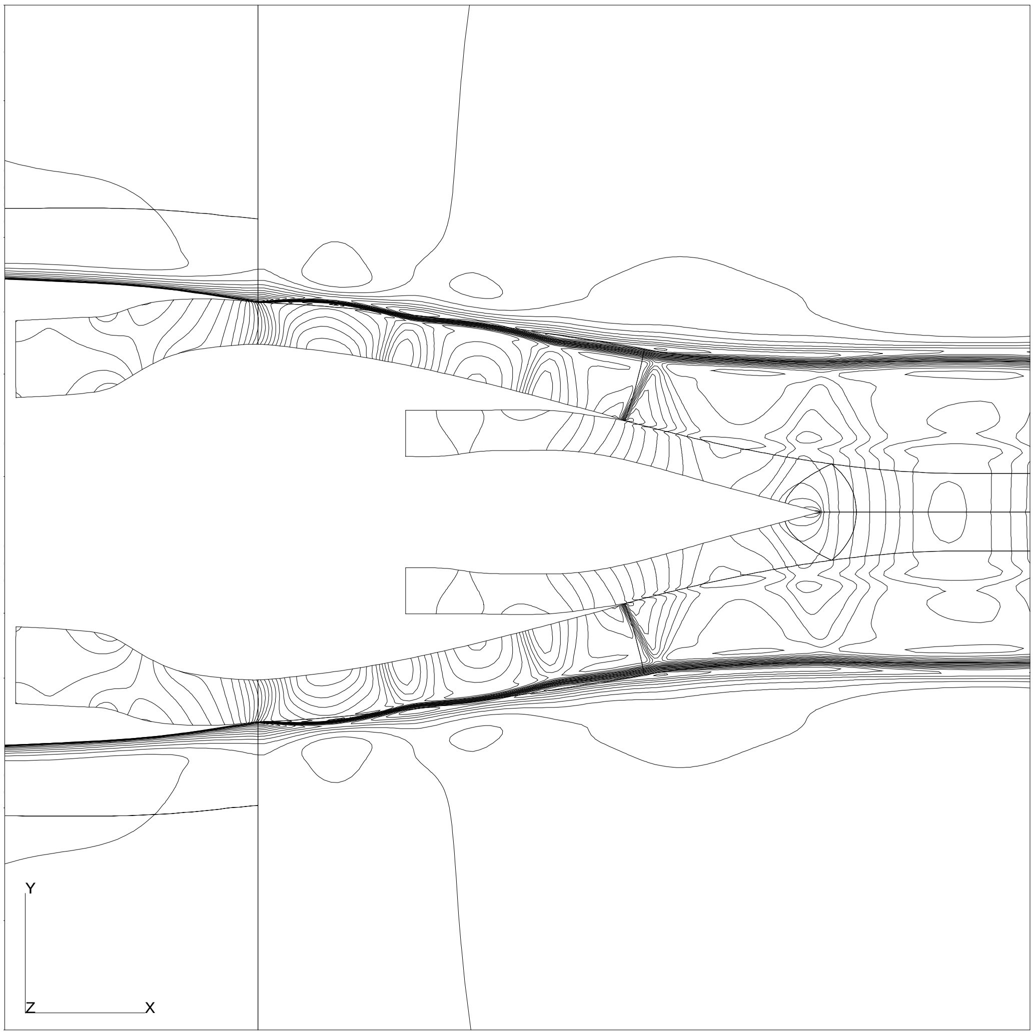

Figure 4: Lines of constant Mach number in the vertical symmetry plane in outlet area. Axi-symmetric flow. Fine grid. Contour lines 0.05 in Mach number apart. Observe the rapid expansion (through sonic conditions) of the flow at the outer-jet exit, the three shock waves in the outer jet downstream of this exit, and the velocity-jump shear layer between the flow (subsonic) around the nacelle and the flow in outer jet (supersonic). Further, the traces of a temperature-jump shear layer between the outer jet (which was in this case a cold jet) and the inner jet (which was in this case a hot jet) are visible.

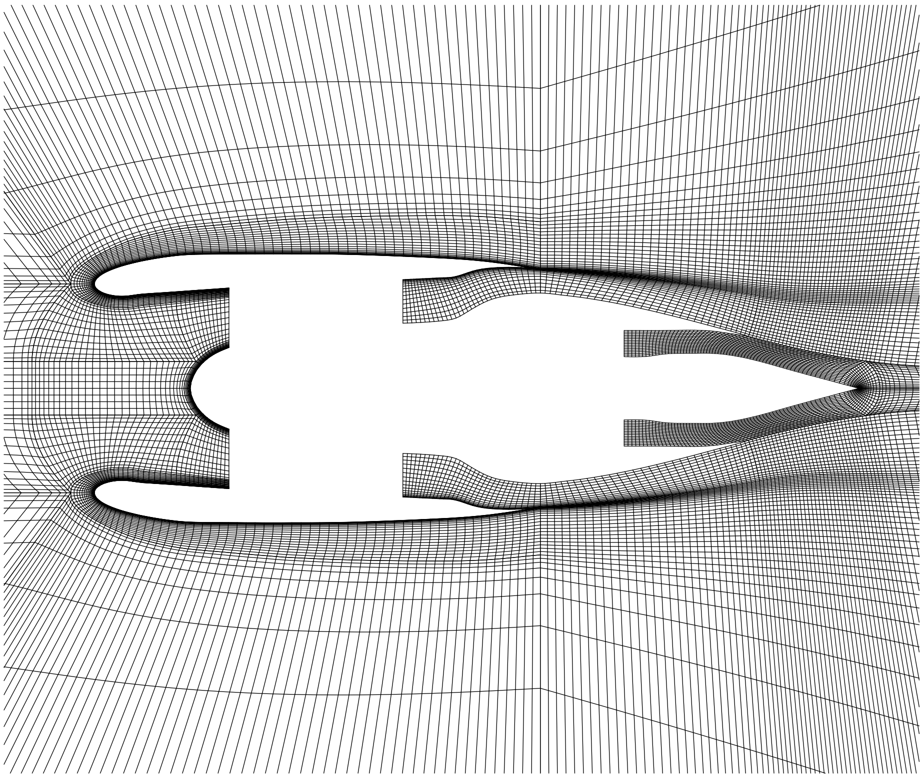

Figure 5: Overview of the medium Navier-Stokes grid in the symmetry plane near engine and nacelle, around the complete engine and nacelle.

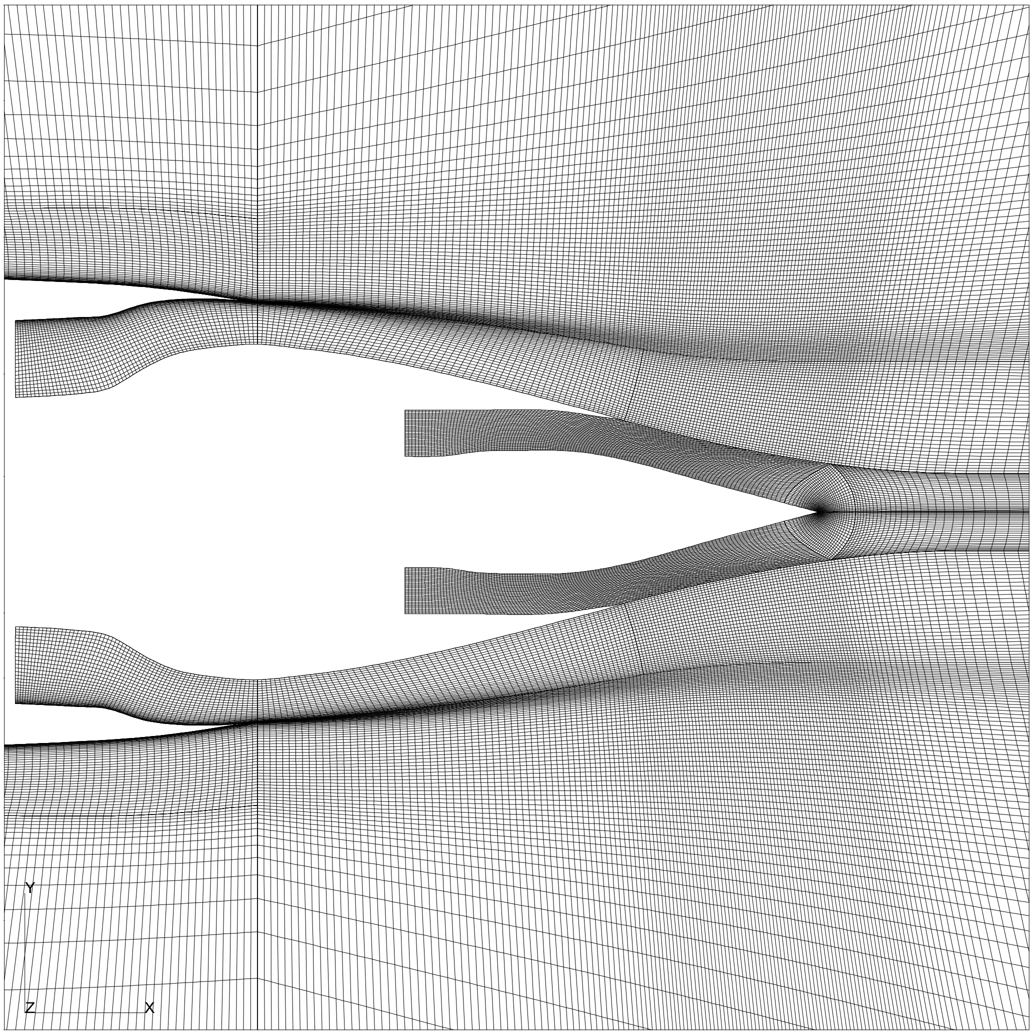

Figure 6: Detail in outlet area of fine grid in the vertical symmetry plane.