Calculation of the viscous flow around ship hulls

PARNASSOS Software package

PARNASSOS is a CFD program to calculate the steady incompressible viscous flow around a ship hull, at model scale or full scale. It provides detailed information on the velocity and pressure field around the hull, the wake field in the propeller plane, the possible occurrence of flow separation and the viscous resistance. PARNASSOS is an essential tool for efficiently optimising hull forms for viscous resistance or propeller inflow. It is the result of still ongoing research at MARIN and has been validated against model test and full-scale results.

Applications

Hull resistance and propeller inflow are dominated by viscous flow phenomena, in particular for full-block ships. The flow around the afterbody is the result of the growth and change of structure of the boundary layer along the hull, details of which can only be given by computational programs. Visualisation of the computed flow gives insight into the cause and effect relations. Thus the origin of certain undesired features of the viscous flow field can often be traced back to hull form details, which may be adjusted subsequently. The generation of longitudinal vortices along the afterbody and its effect on the propeller inflow can be computed, allowing for thrust deduction fraction and effective wake distribution. Flow separation may occur on fuller afterbodies, upstream of the propeller, or above afterbodies due to the propeller operation.

PARNASSOS is an efficient tool to calculate these viscous flow phenomena, and gives detailed information on changes from model to full scale in a straightforward manner. PARNASSOS is one of the very few commercial CFD codes, commercially available that is able to predict the full-scale viscous flow without using wall functions and with a high numerical accuracy. Calculations generally are done for zero drift angle and symmetric flow. Depending on the goal of the calculation the free surface is flat (so-called double body flow) or it can be modelled by inserting a result from a free surface panel code, like RAPID. PARNASSOS is optimised for so-called bare hull calculations, where skegs, tunnels, and headbox can also be modelled. The degree of geometrical complexity to which the code can compute, is part of the ongoing development and is still increasing (see http://www.marin.nl/web/Facilities-Tools/CFD/PARNASSOS.htm).

Computational approach

PARNASSOS solves the Reynolds-averaged Navier-Stokes (RANS) equations in a body-fitted curvilinear grid, using a finite-difference method. Various turbulence models have been implemented, varying in complexity. Depending on the calculation different models can be used.

The near-wall flow is accurately resolved by using grid refinement to the wall and solving the RANS-equations up to the wall instead of modelling the near-wall flow via a coarse near-wall grid and approximate wall functions. The solution is obtained in an efficient multiple-sweep marching process that takes maximum advantage of the existence of a predominant flow direction. The effect of the propeller on the flow can be modelled by a force field representation. The code’s efficiency permits to use dense grids (more than a million cells for a complete ship) and thereby reach a high numerical accuracy. Validations have shown a generally good prediction of the flow and separation phenomena. Wakefield predictions do not always match experimental data, due to the well-known limitations of current turbulence models.
A typical calculation can be done on a standard PC within one day, making it very suitable for hull optimisation studies.

**User interface**

The geometry of the hull is described as a B-spline surface, either created from line drawings or imported digitally (IGES) from a standard CAD-package. In a first graphical user interface the wall mesh on the ship’s hull is made; this user interface can be an integrated part of the MARIN CAD-package GMS. In a second user-interface a boundary-fitted mesh is generated in a domain around the hull. The quality of this mesh can be optimised interactively by manipulation of several control functions. Outer boundary conditions for the calculation are derived from a potential flow calculation. For a calculation for an afterbody only, an inflow plane halfway along the ship is selected where inflow data are generated automatically based on thin boundary layer formulas. Of course also full ship calculations can be done.

The velocity field and pressure distribution in the entire domain are predicted and can be visualised using the visualiser Tecplot. Limiting streamlines on the hull (a ‘numerical paint test’) show the location and strength of any flow separation. Detailed information on the velocities in the propeller plane can be extracted.

One can easily extract tracing streamlines from the computed flow field which indicate how flow features affect the flow downstream.

**References**


For more information please contact the department
Maritime Simulation & Software Group;
T +31 317 49 32 37
E msg@marin.nl