ABOUT AERODYNAMIC CALCULUS OF IMPELLERS USING THE NUMERICAL METHODS

Beazit ALI*, **Gheorghe SAMOILESCU***

*"Mircea cel Bătrân" Naval Academy, Constanța, Romania

***Abstract**: An algorithm for aerodynamic calculus of impeller using numerical methods (Computational Fluid Dynamics) is presented. The geometry of the impeller was analytically determined and the main objective is to compare the analytical solutions of lift with the numerical solutions.*

***Keywords**: impeller, numerical solution, analytical solution, propeller.*

1. INTRODUCTION

A Hovercraft is also sometime called an Air Cushion Vehicle or ACV. This is a vehicle that can drive on land like a car but will traverse ditches and small gullies like it is flat terrain. The Hovercraft is a unique method of transportation.

Modern Hovercraft are used for many applications where people or equipment need to travel at speed over water but be able to load and unload on land.

The hovercraft engine provides the power to drive fans that blow air under the craft. The air is retained by a rubber 'skirt' that enables the craft to travel over a wide range of terrain. The skirt simply gives way when an obstacle is encountered.

The engine also supplies power to a thrust propeller that pushes the craft forward on its 'bubble' of air. Rudders, like on an airplane, steer the direction of the craft. The propeller used to impeller for to drive the hovercraft along is usually an aircraft type with fixed or variable pitch blades.

For the analytical solution of thrust many hypotheses are made. Aerodynamic calculus of impeller's propeller using numerical methods (Computational Fluid Dynamics - CFD) can give us more accurate solutions for thrust. Also the distribution for the parameters of aerodynamic field is determined.

2. PROBLEM DESCRIPTION

The problem consists in the flow through a hovercraft fan with 6 blades. Due to cyclic periodicity only one blade will be modeled.

For geometric model was used GAMBIT software. For each aerodynamic profile were introduced 33 points (10 sections). The geometry of the impeller's propeller is the one determined in the analytical calculus. Figure 1 present the geometry of the fan and the boundary conditions.

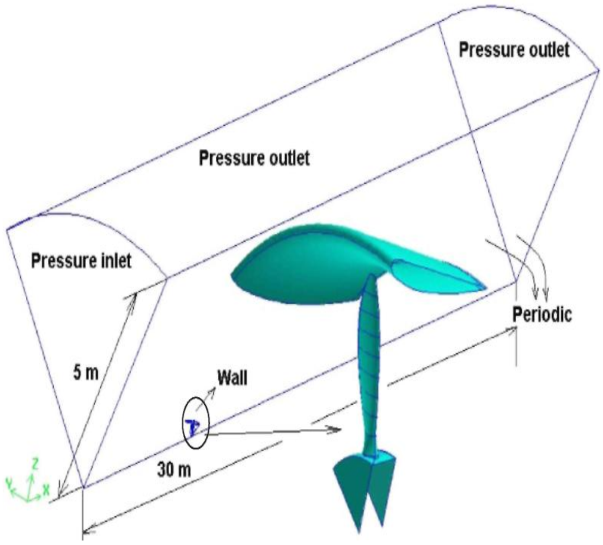

Fig. 1 Geometry and boundary conditions

As shown in figure 1, domain's extremities were chosen far enough from the fan.

For the lateral faces of the domain cyclic periodicity condition was applied.

The domain is rotating at the corresponding speed (fan speed – different values for each case considered) and because of this the ring wall has only the no-slip condition (implicit for turbulent flows) and the speed of the blade wall was set to 0 m/s.

For operating conditions the pressure was set to 101325 Pa.

In consequence, the boundary conditions are:

- "wall" for blade – Stationary Wall;
- "wall" for ring – Moving Wall – Relative to adjacent cell zone – Speed 0 rot/sec – Rotational – Direction (1,0,0) (x axis);
- "fluid"– Moving reference frame - Speed n rot/sec – Rotational – Direction (1,0,0) (x axis)
- "pressure outlet" – Gauge pressure 0 Pa – Backflow turbulence intensity 0,05% - Backflow turbulence viscosity ratio 1;
- "pressure inlet" – Gauge pressure 0 Pa – Turbulence intensity 0,05% - Turbulence viscosity ratio 1;
- "periodic" – rotational.

3. NUMERICAL SOLUTION

The discretisation of the domain was made considering a finer mesh around the blade and the ring, where the gradients are bigger and a coarse mesh at extremities. The final mesh with 524 000 tetrahedral cells is presented in figure 2.

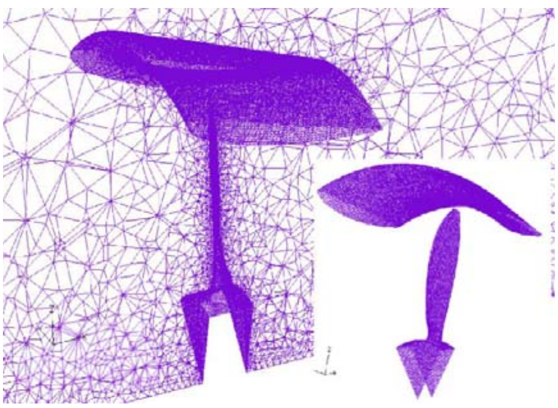

Fig. 2 Unstructured grid – 524 000 tetrahedral cells

The lateral faces of the domain were link-meshed for mesh correspondence in cyclic periodicity.

The working fluid is air with standard proprieties.

The solutions were determined using the segregated solver, implicit formulation. The implicit formulation has a faster convergence but needs more computational resources. The segregated method solves Navier-Stokes equations separated using the algorithm presented in figure 3.

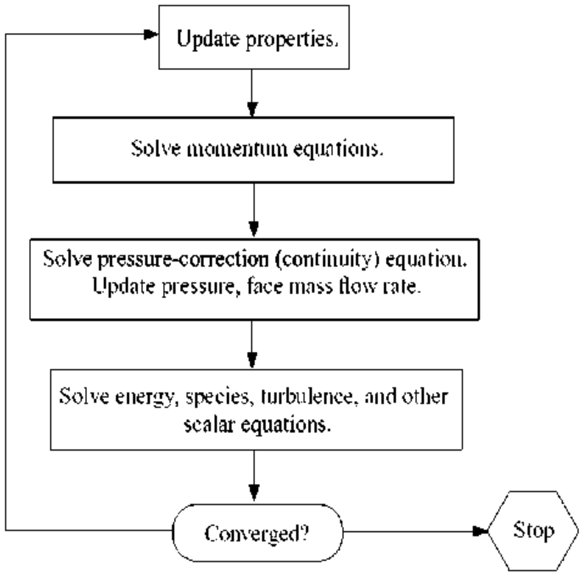

Fig. 3 Segregated method

The turbulence model is k-ε standard model and for each solution (rotational speed of the fan) were made 1000 - 1500 iterations until convergence.

The numerical solution was calculated for:

Z = 6 – number of rotor blades;

D = 1,15 m – diameter of rotor;

P = 20 kW; 30 kW; 40 kW; 50 kW; 60 kW – engine power;

n = 2000 rot/min; 2500 rot/min; 3000 rot/min; 3500 rot/min; 4000 rot/min – rotational speed of rotor.

In figure 4 is presented the distribution of static pressure for P = 60 kW, n = 3500 rot/min.

As was mentioned before, the reference pressure is 101 325 Pa.

In figure 5 is presented the distribution of velocity for P = 60 kW, n = 3500 rot/min.

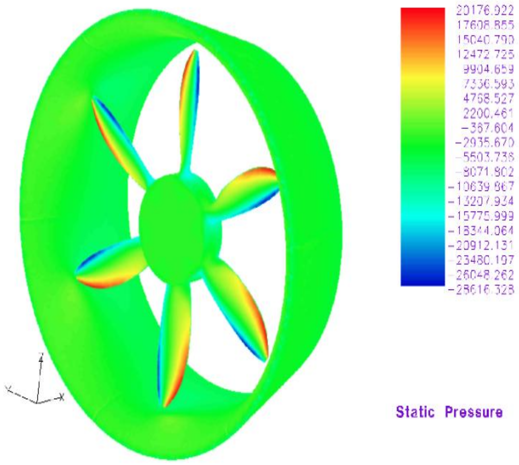

Fig. 4 Distribution of static pressure

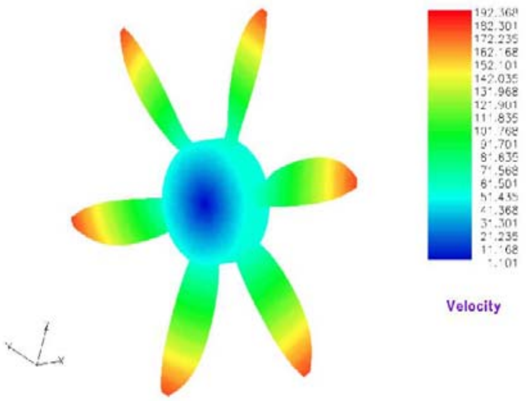

Fig. 5 Distribution of velocity

4. RESULTS AND DISCUSSIONS

The blade geometrical parameters listed in table 1 and 2 were determined in the analytical calculus.

In tables 3 and 4 are listed the analytical and numerical values calculated for thrust. There are also calculated the relative errors between analytical and numerical solutions.

Relative error = (numerical value – analytical value)/analytical value.

Table 1 Distribution of chord length (c) and airfoil width (b) along R

Type	I	
r b/c	b	c
30%	46,1	87,41
40%	18,99	109,1
50%	14,84	121,55
60%	13,02	128,58
70%	11,82	128,49
80%	10,22	117,55
90%	7,39	87,91
100%	3,73	45,16

Table 2 Reference angle at r = 0,75 (D/2) [°]

n(rot/min) P(W)	2000	2500	3000	3500	4000
20000	16.06	10.17	6.62	4.20	2.41
30000	20.86	13.49	9.17	6.29	4.22
40000	25.09	16.31	11.29	8.01	5.68
50000	29.00	18.84	13.16	9.50	6.93
60000	32.74	21.18	14.86	10.85	8.04

Table 3 Relative errors between analytical and numerical solutions. Thrust [N]

P\n	2000 rpm			2500 rpm			3000 rpm		
	Analytic	Numeric	Relative error	Analytic	Numeric	Relative error	Analytic	Numeric	Relative error
20000	837.3	874.26	+ 4.4 %	803.2	851.3	+ 5.9 %	721.4	789.2	+ 9.4 %
30000	1096.2	1250.2	+ 14 %	1088.5	1245.6	+ 14.4 %	1034.5	1215.2	+ 17.5 %
40000	1316.5	1489.1	+ 13.1 %	1329.5	1426.1	+ 7.2 %	1295.3	1425.6	+ 10 %
50000	1510.6	1725.3	+ 14.2 %	1543.5	1759.5	+ 13.9 %	1526	1746.1	+ 14.4 %
60000	1686.7	1856.8	+ 10 %	1739.4	1902.7	+ 9.3 %	1735.5	1923.1	+ 10.8 %

Table 4 Relative errors between analytical and numerical solutions. Thrust [N]

P\n	3500 rpm			4000 rpm		
	Analytic	Numeric	Relative error	Analytic	Numeric	Relative error
20000	578	642.8	+ 11 %	358.7	425.6	+ 18.7 %
30000	925.5	1098.4	+18.7 %	758.4	889.4	+ 17.2 %
40000	1213.3	1459.3	+ 20.2 %	1078.2	1215.3	+ 12.7 %
50000	1462.2	1614.1	+ 10.3 %	1350.2	1498.7	+ 10.9 %
60000	1690.1	1923.2	+ 13.8 %	1591	1775.2	+ 11.5 %

5. CONCLUSIONS

The model was generated using the coordinates determined from analytical solution and respects exactly the aerodynamic surface. The accuracy of aerodynamic surfaces is mandatory for CFD analyses.

Parametrical definition of geometry allows us to change very easy the problem.

The numerical values are bigger than the analytical values because in calculus of the numerical solution the flow is turbulent and the propeller is placed inside the ring.

The thrust of tubed propellers can't be calculated exactly with analytical algorithms. There are experimentally determined coefficients witch multiplies the thrust of free propeller. In some references the increasing in thrust is maximum 30%, depending of the shape of the rings.

REFERENCES

1. Dumitrescu, H., *Calculul elicei*, Editura Academiei Române, Bucureşti, 1990;
2. *** *Fluent – Documentation*;
3. *** *Gambit – Documentation*;
4. *** *Visual Basic for Applications – Documentation.*