# CFD SIMULATION OF DYNAMIC LIFT ON AIRFOIL

**Peter GAŠPAROVIČ**[1]

***Abstract:*** *Rapidly rotating airfoil is subject to dynamic lift. Dynamic lift can cause excess stress on aircraft structure. There are few data on this phenomenon. This article presents CFD (Computational Fluid Dynamics) results of dynamic lift ramp-up test on simple airfoil at various angular velocities.*

***Keywords:*** *dynamic lift, dynamic stall, airfoil, CFD*

## 1. INTRODUCTION

Dynamic lift is frequently occurring phenomenon on rotor blades of helicopters and windturbines. However it can be rarely encountered on propeller blade in strong crossflow or even on wing of rapidly manoeuvring aircraft. Dynamic lift can occur when streamlined body is in rotational movement and its maximum deflection exceeds usual range of angles of incidence. Usual range of incidence is limited by stationary stall. Stall is the state characterised by maximum values of normal aerodynamic forces and is therefore one of the cases for structural design of flying transport vehicles. Dynamic lift may lead to several times larger magnitude of normal forces compared to stationary stall, therefore if not anticipated in design stage, it can be vary dangerous in operation. However the proper treatment of dynamic lift in design stage is very difficult, because the causes of dynamic stall are more complicated and prediction of dynamic stall taking experimental and computational tools is more expensive. One of the tools is Computational Fluid Dynamics (CFD), and this article presents preliminary result of use CFD system Fluent for prediction of dynamic lift.

## 2. DESCRIPTION OF CFD SOLUTION

The core of CFD system Fluent is numerical iterative solver. It solves equations describing the airflow of model case by conservation of momentum, mass and energy. Geometric model of the flow must be discretised in the form of computational mesh, and equations are solved on this mesh subsequently. As a test object, the NACA 0012 airfoil was selected, because its stationary characteristics are well known. Test case of numerical simulation is described as stationary setting airfoil at 0° angle of attack, and then quickly rotating it at constant angular speed (ramp-up) up to 90° angle of attack. Centre of rotation was located at first quarter of chord. Tree angular speeds were simulated: 30, 90 and 360°/s. They are higher than usual speeds used in most of experiments. They can be expressed in nondimensional form of *reduced pitch speed* (angular speed x chord / airspeed) at 0.037, 0.11 and 0.44 respectively. Airspeed was set at 14.5 m/s, which leads to Reynolds number of 1 million for airfoil chord of length 1 m.

The geometric model consist of circular domain with radius 20 times chord of airfoil, which is 1 m long and is located in the middle of domain. The geometry was meshed using the software ICEM CFD. The mesh is based on structured O-grid topology (fig. 1) and properly resolves boundary layer, i.e. the size of cells adjacent to walls is limited by $y^+ = 1$ (fig. 2). Resulting size of the mesh is 45934 cells and 46320 nodes.

The computation was initialised by steady state solution at 0° angle of attack. Then the reference frame of computational domain was set into rotational motion about point in first quarter of the chord, and solver was switched to unsteady calculation. The speed of rotation was constant during whole motion from 0° to 90°. The first small timesteps (10 steps / 1°) were conducted in order to dissolve effects of unphysical infinite acceleration of rotation. Other computations were performed at rate 1 timestep / 1°. The lift coefficient was calculated at each timestep.

---

1

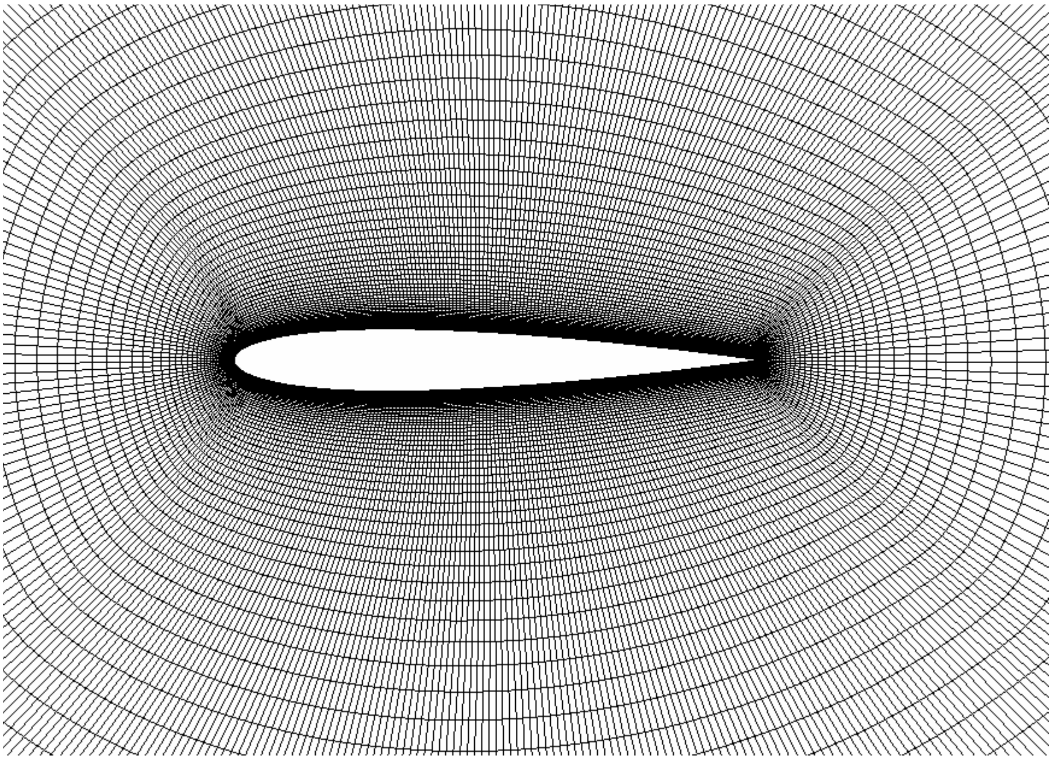

*Figure 1 O-grid topology of mesh*

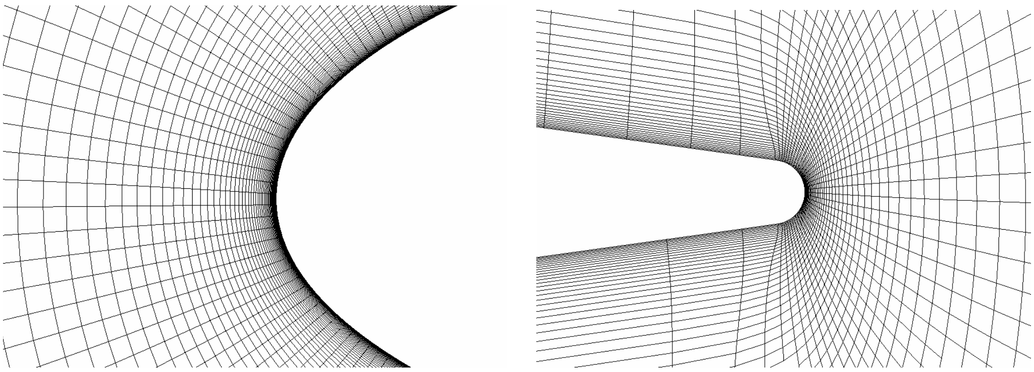

*Figure 2 Detail view of boundary layer of the mesh*

## 3. RESULTS AND DISCUSSION

Collected lift curves are showed in fig. 3. It is clear, that there exist constant step at the beginning of curve. This step is proportional to angular speed and is caused by enlarged circulation because of airfoil motion. Slope of lift curve for dynamic cases is lower than for steady state case. It can be explained by shedding circulation in the wake because of time-dependent behaviour. Maximum lift is function of angular velocity. Because movement is not of oscillating but ramp-up type, there is no reason to expect dynamic stall. The decrease of lift behind 45–50° angle is contributed to change in geometry, where the projected area of airfoil in the direction of lift is decreasing to zero as it approaches 90° angle of attack.

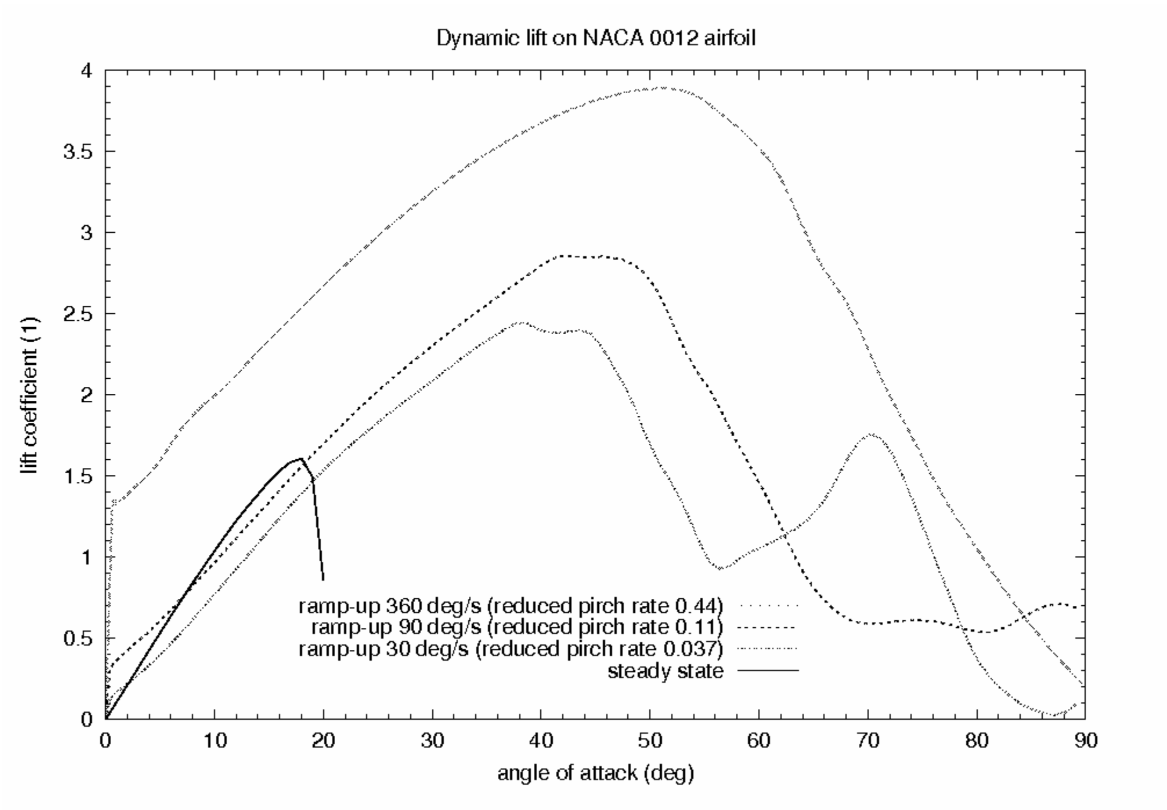


*Figure 3   Lift curves at various angular velocities*

## 4. CONCLUSION

Presented method allows to predict dynamic behaviour of airflow around rotating airfoil. For precise prediction of dynamic stall it is further needed to investigate ability of the method to model viscous effects on the time-dependent airflow around rotating airfoil.

## REFERENCES

1. ROZEHNAL Dalibor: *Wind tunnel testing of stacionary and nonstacionary aerodynamics characteristics of the airfoil*, In: Zborník 6. medzinárodnej konferencie Nové trendy v rozvoji letectva, Košice, Slovak Republic, 2004, pp. 82-88
2. Sheng, W. and Galbraith, R. A. McD. and Coton, F. N.: *A new stall onset criterion for low speed dynamic stall,* Journal of Solar Energy Engineering 128(4), 2006, pp. 461-471, url: http://eprints.gla.ac.uk/3381/

**Opponent:** prof. Ing. Jozef Považan, CSc., Jozef.Povazan@tuke.sk, Faculty of Aeronautics, TUKE, Rampova 7, Kosice, SLOVAKIA, phone: 0918 691 172