Previously, computational fluid dynamics (CFD) computations of dynamic stall on wind turbine blades have been performed for stand still conditions with moderate success by among others the present authors, in the present work, numerical investigations are performed to illustrate the possibilities of state of the art CFD methods for this problem, including the numerical requirements as time-step and grid resolution. Additionally, the effect of different types of modeling is investigated, ranging from fully turbulent Reynolds-averaged Navier-Stokes (RANS), transitional RANS, to transitional delayed detached-eddy simulation computations. The investigation indicates that detailed information and fair agreement with measurements can be obtained.