

Rotating flow instability prediction using eigenvalue analysis

Shenren Xu ^{*1}

¹*School of Power and Energy, Northwestern Polytechnical University, Xi'an, 710072, China*

The compression system in turbomachines, e.g., aircraft engines and gas turbines, when operating under off-design conditions, exhibits self-excited unsteady phenomena such as surge, rotating stall and rotating instability, leading to performance deterioration and/or structural damages. Inability to accurately predict when such flow instability occurs limits the development of high performance compression system. In this paper, an eigenvalue problem is solved to (1) predict the linear stability and (2) find the destabilizing eigenmode for a classical commonly seen for turbomachines operating at near stall conditions. The eigenvalue analysis is fully based on the steady state three-dimensional Reynolds-averaged Navier-Stokes equations and thus the stability boundary is fully consistent with the one that is predicted by the time-accurate flow simulation, i.e., URANS, but two to three times faster. The method is applied to the computation of stability boundary of (i) the laminar flow around a two-dimensional circular cylinder, (ii) the flow around a quasi-three-dimensional compressor annular cascade, and (iii) flow around a three-dimensional compressor rotor. The method developed here has the potential to revive the once-popular eigenvalue method for prediction rotating stall and surge, which was based on lower-fidelity flow models and provide industry with tools to accurately predict the stall line in the early design stage.

^{*}Corresponding author, email address: shenren_xu@nwpu.edu.cn

I. The nonlinear flow solver

A. Governing equations

B. Spatial discretization

II. Stability analysis

A. Time-domain unsteady approach

B. Eigenvalue approach

III. Eigenvalue analysis for large sparse matrices

IV. Results

A. Laminar flow around a two-dimensional circular cylinder

Eigenvalue analysis is performed for the canonical case of the laminar flow around a circular cylinder with the Reynolds number in the range between 40 and 100. The computational domain is a circular cylinder centered at the origin with a diameter of $D = 10^{-5}$ and the farfield is a circle with a diameter of $100D$. The left half of the outer circle is set to 'farfield' boundary condition with a incoming flow of Mach 0.2 in the x-direction, a static pressure of 101325 Pa and a temperature of 288.15 K . The right half of the circle is set to 'pressure-outlet' boundary condition, with a constant pressure of 101325 Pa . The computational domain is meshed with quadrilateral elements, with a total of 29600 grid points. The density is 1.225 kg/m^3 . The dynamic viscosity is varied in order to achieve a particular Reynolds number.

1. Steady state calculation

The steady state flow is obtained by either using an implicit solution method in Fluent (version 19.2) or by resorting to the Newton-Krylov algorithm in NutsCFD, despite the fact that the flow is physically unsteady under this condition. The Mach number contour of the NutsCFD calculation is shown in Fig. 1.

To compare the Fluent and NutsCFD results quantitatively, the velocity-x behind the cylinder as well as the pressure coefficient along the cylinder surface are compared in Fig. 2 and very good agreement can be found.

2. Unsteady calculation

Experimental results show that the laminar flow around the cylinder becomes unsteady for Re above a critical value (around 47). To study this phenomenon, unsteady flow calculation for $Re = 55$ and $Re = 40$ are performed using Fluent. First, for both conditions, a steady state flow solution is obtained by converging the residual to machine error. For $Re = 55$, the unsteady simulation is run with the steady state as initial condition. A BDF2 second-order implicit dual-time-stepping method is used with the physical time step set to $10^{-8}sec$, that is, 0.01 ms , and the inner loop is solved

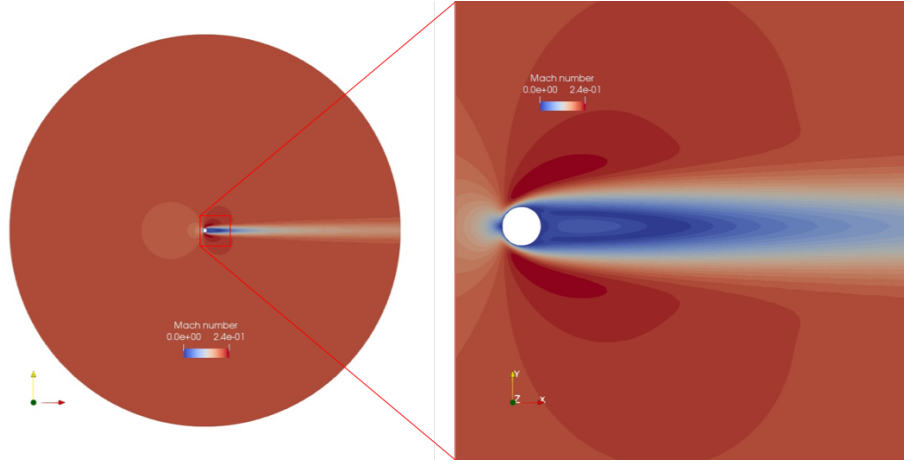


Fig. 1 Mach number contour plot of the calculation results by NutsCFD for $Re = 55$.

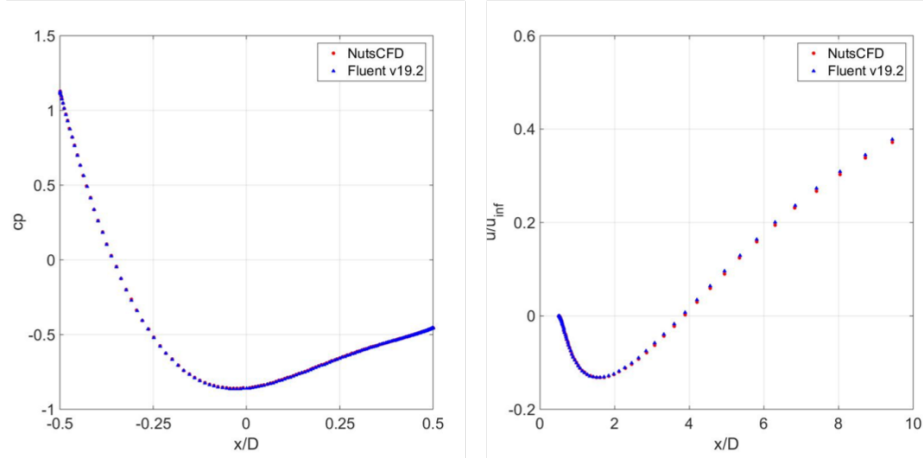


Fig. 2 Comparison between Fluent and NutsCFD calculation results for velocity-x along the center line behind the cylinder (left) and pressure coefficient along the cylinder surface (right).

with a CFL of 1000 and maximum 5 iterations. Roughly two orders of magnitude of residual drop is achieved for the inner loop. From Fig. 3, it can be seen that after around $100ms$, the lift coefficient starts to grow and eventually reaches a saturated limit cycle at around $130ms$. On the contrary, running unsteady simulation with a fully converged steady state for $Re = 40$ does not lead to unsteadiness. To probe the flow at $Re = 40$ further, a disturbance is introduced into the flow from the farfield by setting the incoming flow direction to vertical for one time step and switching it back to the horizontal direction, and then continue the unsteady run. The lift coefficient shows a transient response but eventually slowly delays to zero. These two sets of lift coefficient signals are plotted in Fig. 3 in both linear and logarithm scales. The logarithmic plot on the right clearly shows an exponential growth and decay for $Re = 55$ and $Re = 40$, respectively.

The growth and decay rate for both conditions can be extracted from the time signal and then be compared to the eigenvalue analysis results.

The same unsteady simulation is not performed in NutsCFD as the code is slower than fluent and in the meantime, it is believed that the unsteady response between the two solvers will be very similar, based on the comparison of the steady state solutions.

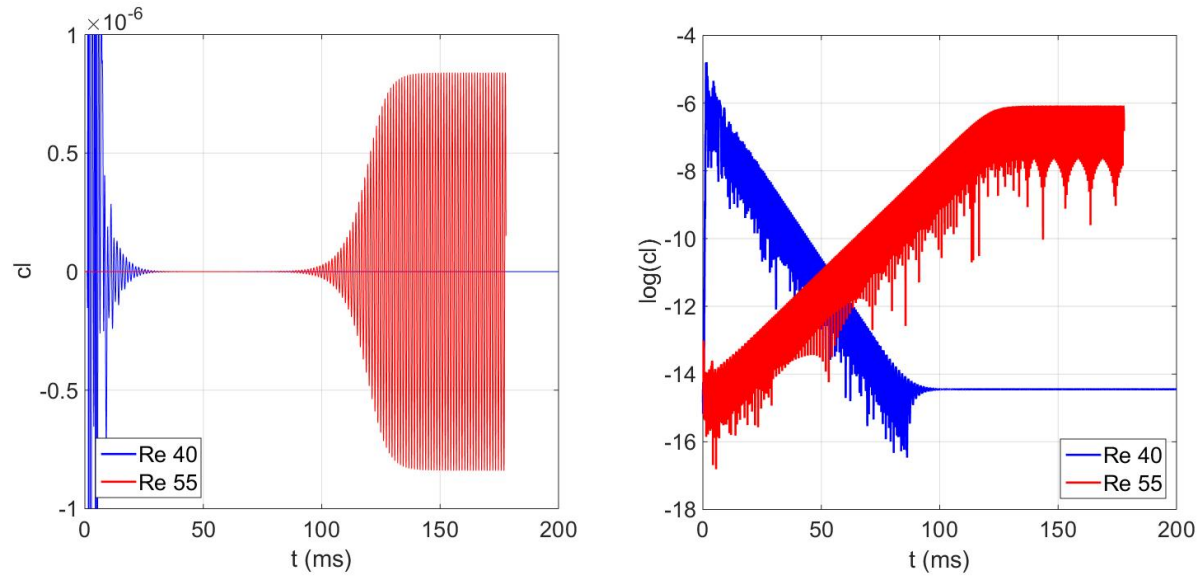


Fig. 3 Lift coefficient histogram for $Re = 55$ and $Re = 40$.

3. Eigenvalue analysis

B. Rotating stall for an axial compressor (NASA Rotor 67)

NutsCFD is used to analyze the performance of the first stage rotor (NASA Rotor 67) of a two stage transonic fan designed and tested at the NASA Glenn center [1]. Its design pressure ratio is 1.63, at a mass flow rate of 33.25 kg/sec. The NASA Rotor 67 has 22 blades with tip radii of 25.7 cm and 24.25 cm at the leading and trailing edge, respectively, and a constant tip clearance of 1.0 mm. The hub to tip radius ratio is 0.375 at the leading edge (TC = 0.6% span) and 0.478 at the trailing edge (TC = 0.75% span). The design rotational speed is 16,043 RPM, and the tip leading edge speed is 429 m/s with a tip relative Mach number of 1.38.

V. Conclusion

Acknowledgements

References

- [1] Strazisar, A. J., Wood, J. R., Hathaway, M. D., and Suder, K. L., "Laser anemometer measurements in a transonic axial-flow fan rotor," 1989.