Ansys + Takushoku University

"Our lab seeks to decipher the biomechanical reason behind animal and plant shapes, particularly from a fluid dynamics perspective. We once tried an open-source CFD software, but it took a few months for students (who may spend less than a year in the lab) to learn it and they were unable to explore the morphological parameter space sufficiently. Therefore, I believe that Ansys Fluent software is one of the best options for students who use CFD as a problem-solving tool."

— **Masateru Maeda**
Associate Professor / Takushoku University

CASE STUDY

Studying the Aerodynamic Effect of Shapes in Organisms with Ansys Fluent Software

Organisms have different shapes for many reasons. One important aspect, particularly for animals who fly or swim, is fluid dynamics. Masateru Maeda, an associate professor at Takushoku University in Japan, is currently working on several research programs with focuses on insect wings, bird feathers, and plant leaves.

The use of numerical simulation proves beneficial for comparing shapes, as it enables users to determine precise geometries through computational models. Completing the study with actual organisms, or individual (intraspecific) variations, would take substantial effort and time. The use of CFD significantly reduces research time and cost.

Challenges

While studying the flight of a dragonfly, the goal was to simultaneously obtain the airflow around the dragonfly wing as well as the strain distribution on the wing (Figure 1) because people now know that a dragonfly wing possesses hundreds of airflow sensors and strain sensors and it is necessary to obtain the precise flow field and strain field so as to study how does the insect utilize the information collected.

Researchers have also worked on the aerodynamics of a bird feather, testing several geometrical models. So far, they have acquired some basic data that indicates that a feather without slits outperforms the one with slits (Figure 2), though there is a need for further research.

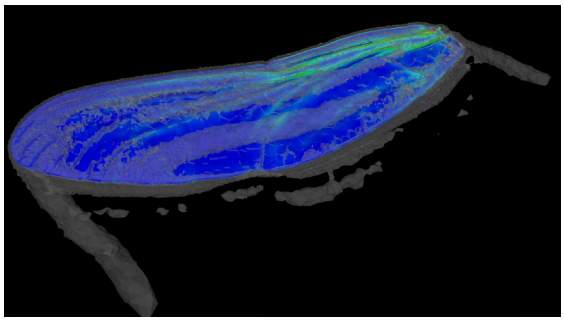

Figure 1. Gliding FSI simulation of a dragonfly wing model showing surface strains and vortical structures (Q-isosurface). This study is in collaboration with Professor Richard Bomphrey, Royal Veterinary College, U.K.

Technology Used

- Ansys Meshing™ 2D/3D mesh generation and analysis capability
- Ansys Fluent® fluid simulation software
- Ansys Mechanical™ structural finite element analysis software
- Ansys System Coupling™ physics solver connection software
- Ansys Workbench™ simulation integration platform
- Ansys DesignModeler™ geometry editing tool
- Ansys CFD-Post capability
- Ansys EnSight™ simulation data visualization software

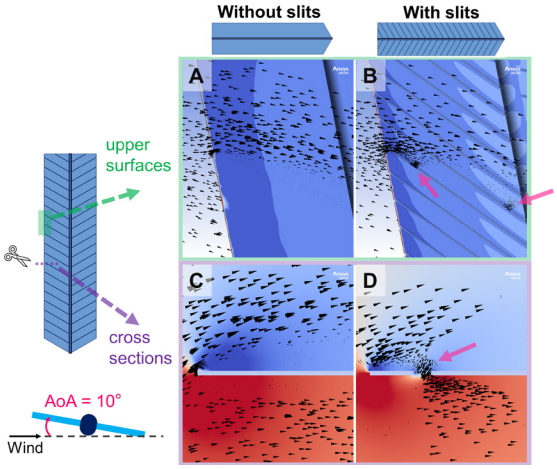

Figure 2. Gliding CFD for two simplified feather models, with and without slits, showing surface pressure (A, B) and cross-sectional pressure (C, D). The black arrowheads detail velocity vectors, while the magenta arrows in B and D indicate flows through the slits.

Engineering Solutions

- Maeda states that the Meshing capability is generally easy and intuitive to work with. He reports it is fairly robust when generating inflation layers (prism layers).
- With Fluent software, intuitive GUI is very effective for beginners. TUI (text user interface) can be tricky, but it is not so difficult once learned and can actually be much more efficient than GUI (e.g., for defining forces).
- Maeda says Mechanical software and the System Coupling tool are impressively easy for running fluid-structure interaction (FSI) simulations once learned.

Benefits

Thanks to the Ansys products, Maeda can focus on the actual simulations rather than the development of numerical methods or generating adequate quality mesh. This is particularly impactful for undergraduate students who have less than a year to complete their thesis projects.

Additionally, fluid-structure interaction (FSI) remains to be difficult in experiments involving living organisms. In numerical simulation, for example, the mass of the distal portion of a dragonfly wing, called 'pterostigma,' which acts to stabilize the oscillation in gliding flight can be instantly changed from the original mass to zero or even 10 times the original mass.

About the Researcher

Maeda received his doctorate from Chiba University in Japan, where he used CFD on insects and hummingbirds with an in-house code written in Fortran 90. He then experienced a few postdocs, including one at the Tokyo Institute of Technology (currently the Institute of Science Tokyo) where he used Ansys Fluent software for the first time to study penguins swimming. Maeda also spent a postdoc at the Royal Veterinary College in London, where he started using System Coupling software for dragonfly wing FSI. In April 2022, he launched a lab at Takushoku University. In the coming years, Maeda would like to expand the variety of organisms studied at the university, including more plants.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724-746-3304
ansysinfo@ansys.com

When visionary companies need to know how their world-changing ideas will perform, they close the gap between design and reality with Ansys simulation. For more than 50 years, Ansys software has enabled innovators across industries to push boundaries by using the predictive power of simulation. From sustainable transportation to advanced semiconductors, from satellite systems to life-saving medical devices, the next great leaps in human advancement will be powered by Ansys.

Visit www.ansys.com for more information.