Life’s Engineering Tales: featuring whale-inspired wind turbines

Facilitator’s Guide

Wen Zhao and Elisabeth Hülse

Ansys Academic Development Team

education@ansys.com

Ansys Software Used

This resource uses Ansys Fluent® fluids simulation software.

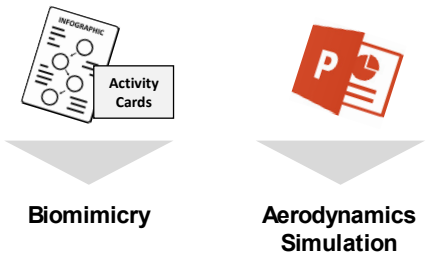
This guide supports the integration of an infographic, a set of activity cards, and a set of slides, into the classroom.

The **poster/infographic** explains how sometimes the great achievements of nature lead to engineering breakthroughs. In this case, a character called “Professor Mimikry” explains how researchers have found that the tubercles on the edge of a humpback whale’s fin enable the whale to take sharp turns and perform agile movements. Engineers were inspired by these findings and applied it to wind turbine blade edges. Simulation shows that wind turbine blades with tubercle edges are more efficient.

The facilitator presenting the poster can walk the participants through these facts, guided by many images and illustrations on the poster.

To add interactive elements to the poster, we have created a set of activity cards and a set of slides.

The activity cards focus more on the topic of biomimicry in general, aiming to inspire to think about other examples. While the slides focus more on the aerodynamic aspects of the topic.



5 activity cards include a question on the front and possible answers on the back for group work. The back also includes a “master mind question” for those motivated to dive deeper.

The **slides** contain simple explanations on aerodynamics and fluid flow in general, what fluid flow simulation does and why it is used, plus how wind turbines generate electricity. The slides also contain questions and activities to engage the kids with the topic.

With this resource we also provide a **simulation file** that can be opened in the Ansys Fluent software, for those who want to run the simulation themselves.

This resource is meant to be flexible in its use. The contents can be used in combination. But using them separately is also possible. Instructors can decide what fits best.

Zip File Contents:

1. Read Me Facilitator’s Guide (this document)
2. Poster/Infographic PDF
3. Activity Cards PDF
4. PPT slides
5. Ansys Fluent software files for wind turbine blade

Age Range: 6-99

This resource is designed to tell a story, give food for thought and inspire knowledgeable discussions. It is worth noting, that we intentionally use simplified language avoiding specific jargon, to make the topic understandable and relatable for children (**age 6+**) and anyone trained outside of this subject area.
The focus is on getting the participants (whether young or old) to talk about STEM topics as a group, no matter at which depth, and think outside the box and be inspired.

Learning Objectives:

After going through this activity, learners will gain a basic understanding of:

- Concept of biomimicry
- Aerodynamics / Fluid flow dynamics
- Basic understanding of Computational Fluid Dynamics (CFD)
- How wind turbines generate electricity
- How understanding science and nature can lead to great innovations

Format Suggestion:

Poster plus activity cards:
We expect it to take approximately up to **90-120 minutes** to go through the poster and add group work using the activity cards. Below is a breakdown of an example of how this resource could be implemented:

1. Facilitator presents the poster, allowing a few questions if desired **(5 min)**
2. Divide participants into up to *5 groups* **(5 min)**
3. Each group gets one activity card and is asked to have a discussion around the question on their card. **(10 min)**
4. Group is allowed to look at answer on the back and can discuss and compare the answers on the card with their previous discussion. For those ahead of time, the master question can be incorporated into this discussion. **(5 min)**

5 minute break

5. Each group presents a summary of their discussion, including challenges and take-home message **(i.e. 5 min per group)**
6. Time for Q&A and open discussion **(5 min)**

Adding the slides:
We expect this to add approximately **45 minutes** to the activity above for the facilitator to present the slides and also use the incorporated questions to the audience.

Please note, this format is a suggestion and can be adjusted by the facilitator as best suitable. It is, for example, also possible to exclude the activity cards and just use the poster to introduce the topic and then move on to the slides.

Below you will also find suggestions for further resources and activities expanding on this topic.

Prior/Supplemental Knowledge Required:

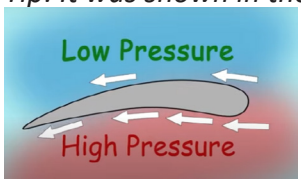
No prior knowledge is required for this exercise.

Fundamental concepts covered:

This case study covers concepts such as fluid flow dynamics, fluid flow simulation, biomimicry, interdisciplinary nature of the field of engineering.

Additional Simple Activities to Expand Thinking on this Topic:

1. Watch the following video with your students: **"What is Lift?"**
   Now, have the kids draw a simple schematic that explains how "lift is generated".
   Tip: It was shown in the video and looks like this:



2. Watch the following video with your students: **"What is Drag?"**
   Try to recreate some activies shown in the video, i.e.:

Additional Resources:

Pre-University Level (ages 6+):

Ansys Everyday Engineering Videos:

- "Why Golf Balls Have Dimples" - 2 min video with a short and simple fluid flow example

Other Ansys pre-university resources:

- "Why this shape? Exploring the historical and structural significance of the Arch part 1"
- "Materials Intelligence: the Card Game"
- "Selecting Materials for Musical Instruments: a Case Example with a Xylophone"
- "Our impact on the planet: let's make it a good one!"
- "Life's Engineering Tales: Grandma's Hip Replacement"
- "Life's Engineering Tales, featuring: An element's journey"
- "Exploring Material Properties via Experiments and Property Charts: the Game"

Advanced Aerodynamics Resources:

Ansys Innovation Courses (AICs) on related topics:

AICs are free, online courses including lecture videos and handouts, followed by hands-on activities and quizzes. In many of the course activites the use of Ansys Software (free student version) is included.

- "Aerodynamics Discovery Day"
- "Exploring the Physics of Drag"
- "Exploring the Physics of Lift"

Ansys case studies on fluid flow:

- "Exploring material properties and physics in Cricket"
- "Effect of Side Spin on a Soccer Ball using Ansys Fluent"

Running Ansys Fluent Software Simulation

Included in this resource download are the relevant files to run the Ansys Fluent software simulation.

We do not include instructions in this resource, as the software is advanced for students at the younger ages targeted.

However, if interested, you can download a free version of the software here

Below are links to some AICs focused on getting started using the Ansys Fluent tool.

- Introduction to Ansys Fluent
- Mesh Generation using Ansys Fluent Watertight Workflow
- Best Practice Guidelines for CFD Simulations

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited. The full Academic Terms & Conditions can be found using this link.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com