



# Read First: Simulation Ready Geometries- Aerospace Collection

Developed and curated by the Ansys Academic Development Team

Alfred Oti

[education@ansys.com](mailto:education@ansys.com)

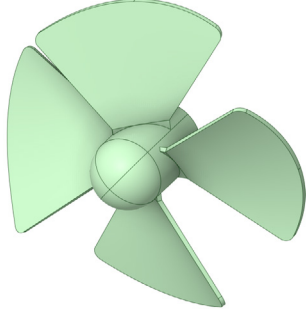
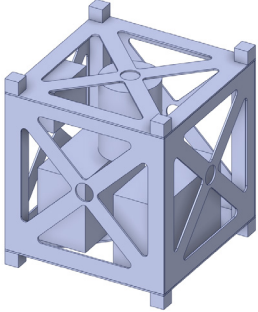
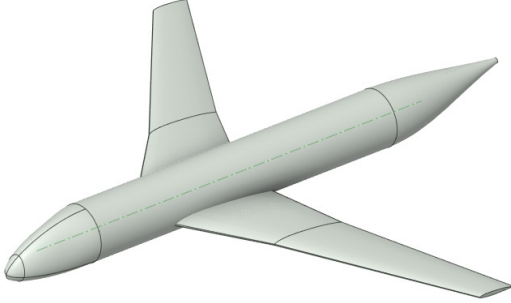
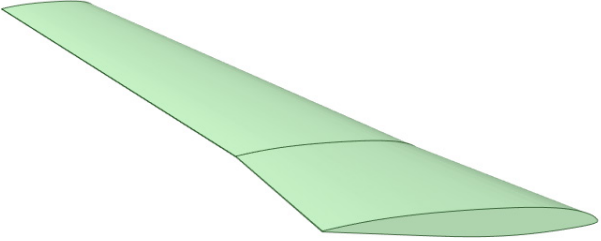
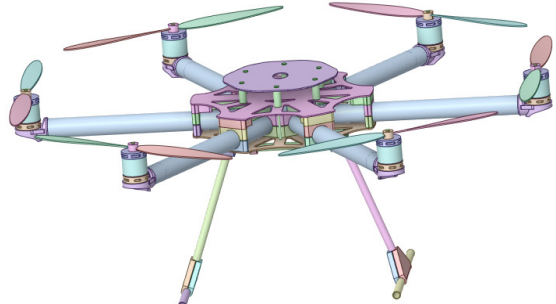
Summary

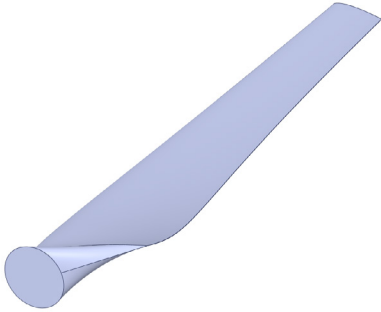
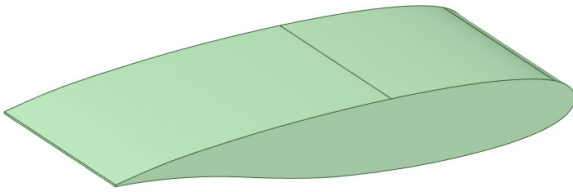
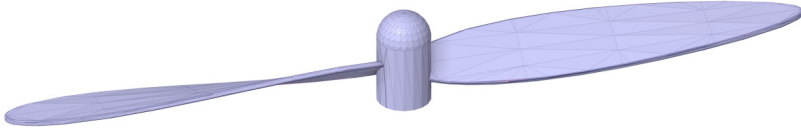
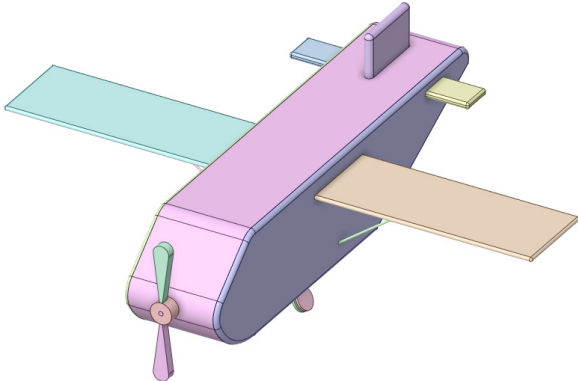
This resource is a collection of Ansys Discovery CAD models, focused on aerospace applications. The goal of this resource is to provide a variety of ready-made CAD models for use in the classroom. Details of the models available, using the Ansys Discovery file format with other Ansys products such as Ansys Workbench, and additional information can be found in this document.


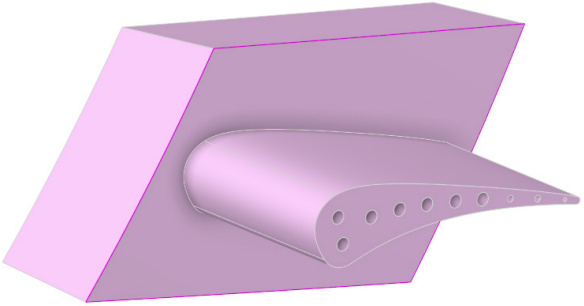
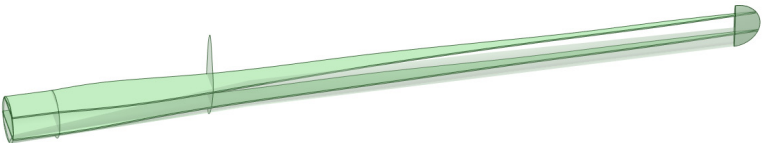
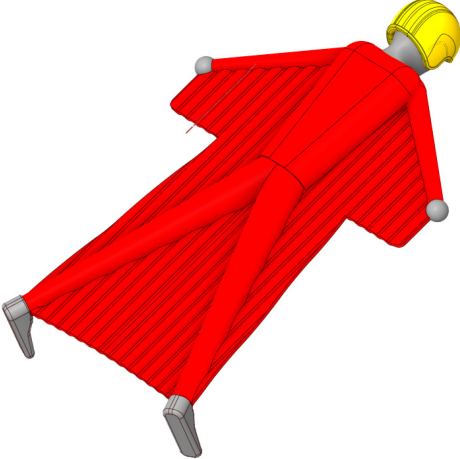
Table of Contents

1. Geometries available in this collection.....	3
2. The .dsco file format.....	6
3. How to use the .dsco file in Ansys Discovery.....	6
4. How to open/insert and transfer .dsco files to Ansys Workbench .....	7
5. Links to Aerospace CAD models .....	8

## 1. Geometries available in this collection

Geometry Name	CAD Image
Axial fan	
CubeSat	
Model airplane	
Radio Controlled Airplane Wing	
Recreational Drone	

Geometry Name	CAD Image
Resurface blade	
Simple airfoil	
Simple drone blade	
Simple lightweight airplane	

Geometry Name	CAD Image
Toy drone blade	
Turbine blade	
Wind turbine blade	
Wingsuit	

## 2. The .dsco file format

In this collection you will find **13** CAD models related to the **aerospace** field. Each model will be available in the **.dsco** file format.



Figure 1: Example CAD file (Disc brake) open in Ansys Discovery

This is the format of the new recommended tool (Ansys Discovery) for geometry editing and preliminary simulation (ready to simulate geometry). Models can be imported into workbench and used with any Ansys software of preference

## 3. How to use the .dsco file in Ansys Discovery

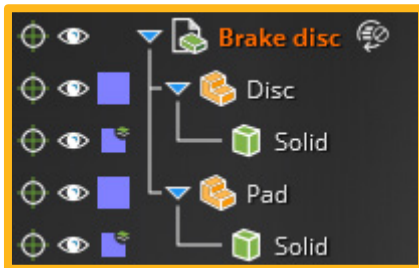


Figure 2: Model Tree in Ansys Discovery for the Disc brake

If any changes need to be made to the .dsco file, Ansys Discovery is the best tool to do this. There are two menus to highlight for quick model changes. We will use the Disc brake from Figure 1 for an example.

When open in Discovery, like in Figure 1, there is a **Model Tree** (Figure 2). This displays the individual parts of the model. For this example, there are two parts held within the containers called **Disc** and **Pad**.

The second drop down menu of importance is the **Physics Tree** (Figure 3). This is one of the places you can change the material used in the model. In this example, we can see cast iron EN GJL 100 is the material currently assigned to the disc brake.

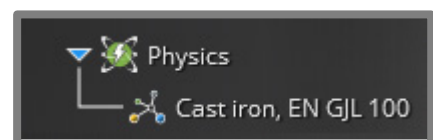


Figure 3: Physics Tree in Ansys Discovery for the Disc brake

## 4. How to open/insert and transfer .dscf files to Ansys Workbench

This section details how .dscf and other supported model formats can be opened in Ansys Discovery and then transferred to flagship Ansys products or Ansys Workbench.

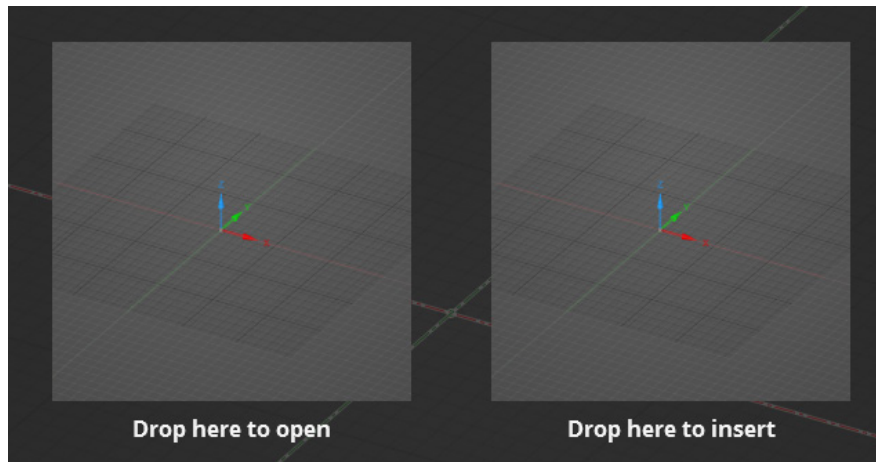


Figure 4: Insert window in Ansys Discovery

.dscf files and other supported model formats can be opened by dragging and dropping files into a blank Ansys Discovery space. Use the Insert window (Figure 4) to include multiple models in the same space.

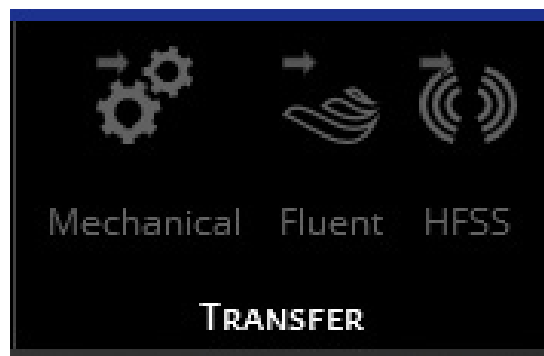


Figure 5: the Transfer Ribbon in Ansys Discovery

To bring .dscf models into the flagship products (Ansys Mechanical, Fluent, HFSS), simply use the Transfer ribbon in Ansys Discovery (Figure 5) and click the product of interest.

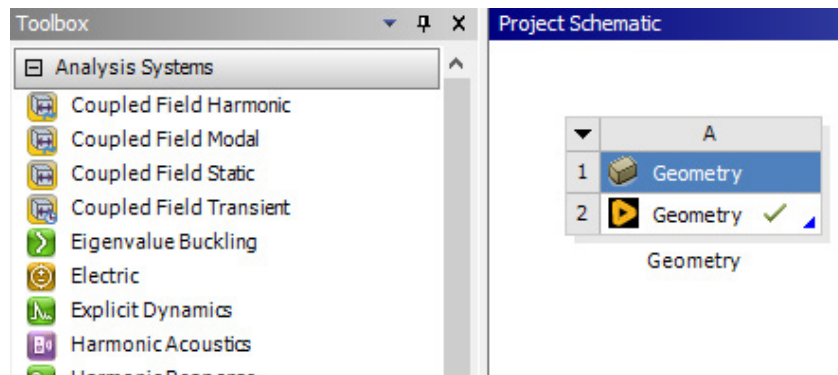


Figure 6: Ansys Workbench geometry file interface

Files can be opened in Ansys Discovery using the Geometry cell in Ansys Workbench (Figure 6).

## 5. Links to Aerospace CAD models

Some of the CAD models included in this package come from sources across Ansys, such as Ansys Innovation Courses (AIC) or the Discovery forum. Links for those sources can be found here.

Model Name	Source Link
Axial fan	<a href="#">Ansys Fluent meshing watertight geometry workflow tutorials-axial fan</a>
CubeSat	<a href="#">Modal analysis of satellite AIC</a>
Model airplane	<a href="#">Ansys Fluent meshing watertight geometry workflow tutorials-generic aircraft geometry</a>
Radio controlled airplane wing	<a href="#">Aerodynamics Discovery Day AIC Lesson 3 Topic Radio Controlled Airplane Wing</a>
Recreational drone	<a href="#">Structures in Aerospace AIC Lesson 2</a>
Resurface blade	<a href="#">Wind blade analysis for wind power using Ansys Fluent AIC Lesson 3</a>
Simple airfoil	<a href="#">Ansys Maker CAD Models</a>
Simple drone blade	<a href="#">Ansys Maker CAD Models</a>
Simple lightweight airplane	<a href="#">Structures in Aerospace AIC Lesson 3</a>
Toy drone blade	<a href="#">Stresses and Local Equilibrium AIC Homework</a>
Turbine blade	<a href="#">Ansys Fluent meshing watertight geometry workflow tutorials-turbine blade</a>
Wind turbine blade	<a href="#">Modal analysis AIC Homework</a>
Wingsuit	<a href="#">Aerodynamics Discovery Day AIC Lesson 3 Topic Wingsuit</a>



© 2023 ANSYS, Inc. All rights reserved.

## 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.

## Document Information

This CAD model collection is part of a set of teaching resources to help introduce students to structures, fluids, or heat transfer (physics areas supported by Ansys Discovery).

## 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](http://www.ansys.com/education-resources).

## Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at [education@ansys.com](mailto:education@ansys.com).

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit [www.ansys.com](http://www.ansys.com) for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2023 ANSYS, Inc. All Rights Reserved.