Manchester Metropolitan University

Creating CAD Geometries in Ansys Discovery Software using 3D Scans

Tutorial 3: Pickleball Paddle

Joshua Hull & Finlay Hare
Supervised by Dr Thomas Allen & Nicholas Hurst.
Manchester Metropolitan University

Reviewed by the Ansys Academic Development Team

© 2024 ANSYS, Inc.

Ansys Software used

Ansys Discovery™ 3D product simulation software is used throughout the different sections of this resource. The Ansys Workbench™ simulation integration platform and Ansys Mechanical™ structural FEA analysis software are also mentioned.

Summary

In this tutorial, we will import 3D scan data of a pickleball paddle into Ansys Discovery software, and convert it into CAD geometry, ready for further analysis. Similar steps are followed to Tutorials 1 and 2.

Figure 1: Pickleball paddle scanned for this tutorial

Table of Contents

© 2024 ANSYS, Inc.

Step 1: Importing the pickleball paddle STL file

Purpose: Import the STL file generated from the 3D scan of the pickleball paddle into Ansys Discovery software, preparing its conversion to CAD geometry (Figure 2).

1. Launch Ansys Workbench software.
2. Navigate to the geometry, right-click, and select “Import Geometry” → “Browse”.
3. Select the provided “Pickleball Paddle” STL file and click “Open”.
4. Alternatively, open Ansys Discovery software, click the “File Menu” button, and select “Open”.
5. Select the “Pickleball Paddle” STL file.

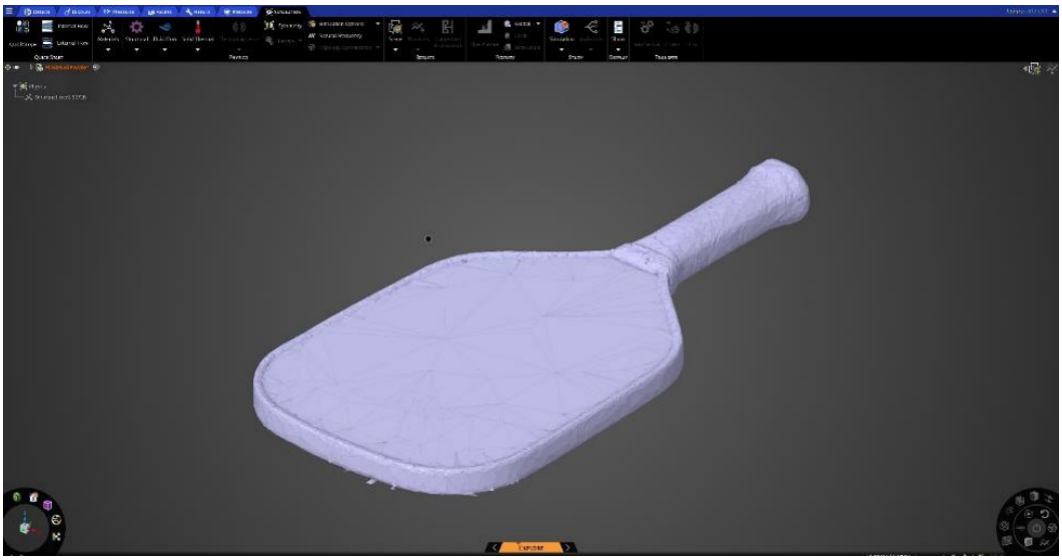

Figure 2: Paddle scan file in Ansys Discovery Software

Step 2: Checking for errors in the pickleball paddle scan data.

Purpose: Identify any errors within the scan data that could prevent CAD geometry conversion.

1. Go to “Facets” in the Ribbon Tab to display the facet tools.
2. Select “Check Facets” tool to identify errors in the scan data, as shown in previous Tutorials. Figure 3 shows the errors.
3. Visualize any errors using the “Holes”, “Intersections”, “Over-Connections, and “Fix Sharps” tool. Use “Auto Fix” to automatically fix errors within the geometry.

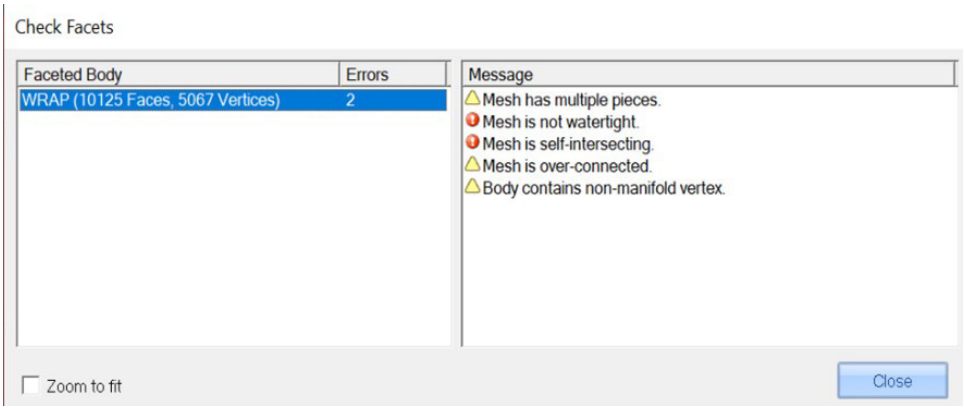

Figure 3: Checking facets

© 2024 ANSYS, Inc.

Step 3: Cleaning up the pickleball paddle scan data.

Purpose: Address and fix any errors within the tennis racket scan data.

1. Use the “Auto Fix” tool to automatically fix errors (Figure 4)

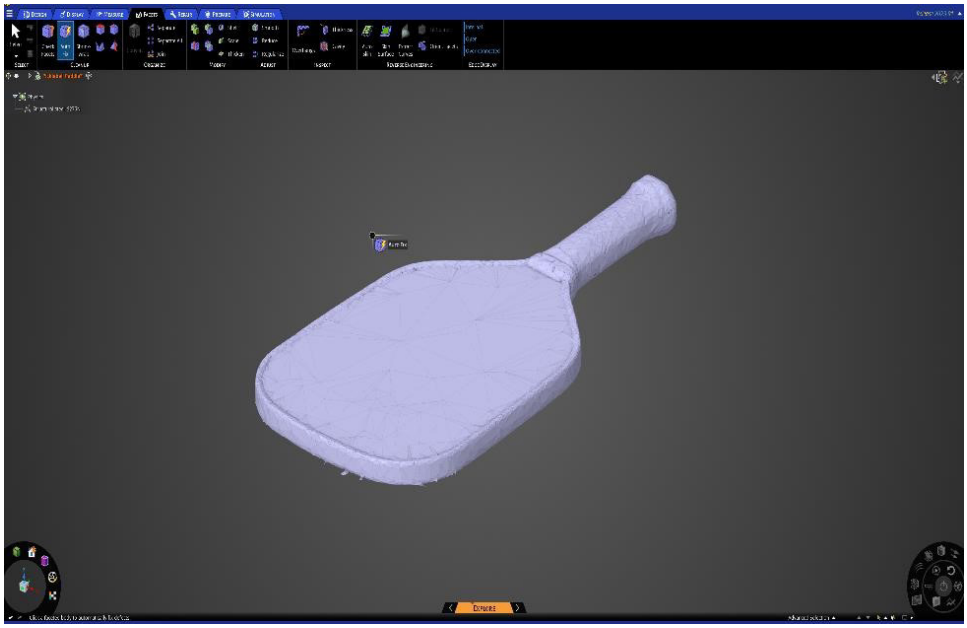

Figure 4: Auto fixing pickleball paddle scan data

2. Select “Check Facets” to check any remaining errors within the scan data (Figure 5)

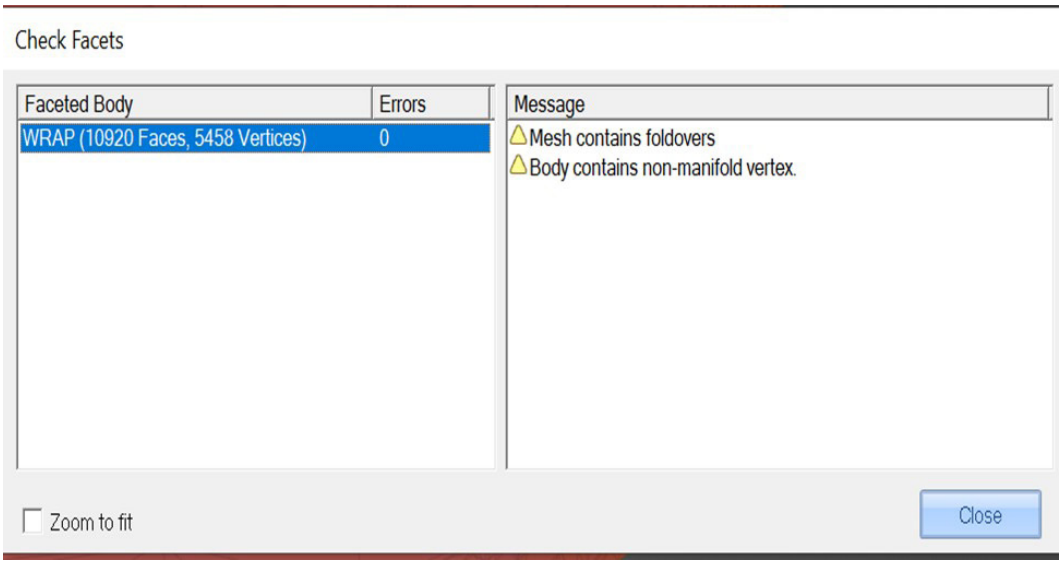

Figure 5: Checking facets after applying the “Auto Fix” to the paddle scan data

3. Further refine the data using “Shrink wrap”, to close any additional gaps or imperfections. Use a size of “1 mm” for the shrink wrap (Figure 6).

© 2024 ANSYS, Inc.

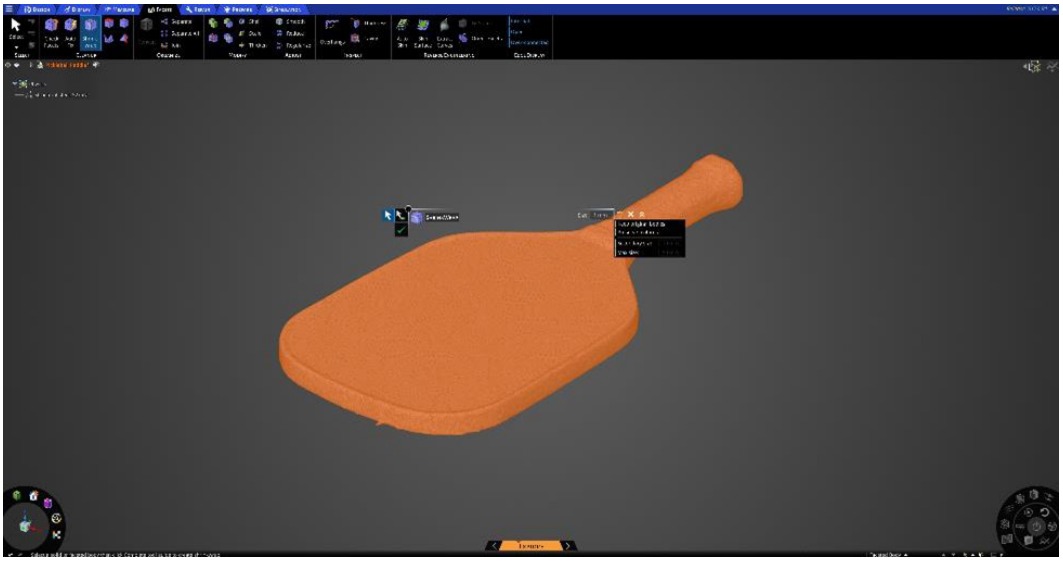

Figure 6: Applying the “Shrink Wrap” to the paddle scan data

4. Use the “Fix Sharps”, “Over-Connected”, “Holes”, and “Intersections” cleanup tools to identify and manually fix any of the remaining errors if necessary.

5. Select “Check Facets” to ensure there are no errors remaining, so the scan data is ready to be converted into CAD geometry (Figure 7).

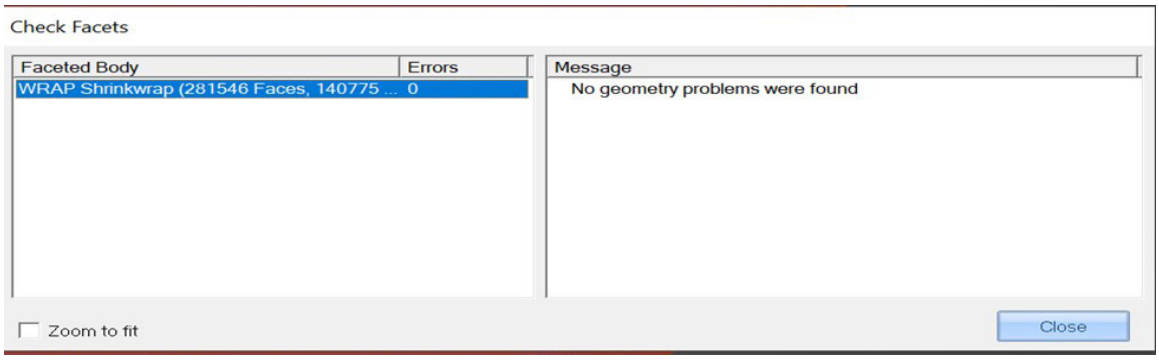

Figure 7: Checking facets after applying the “Shrink Wrap”

Additional Steps:

When moving onto Step 4, if the “Fix Sharp” error message occurs when applying the “Auto Skin” tool to the model, manually fix the errors using the “Fix Sharp” tool while selecting each point individually, or “Shrink wrap” the geometry again. When shrink wrapping, consider experimenting with different size values to find the best results for this geometry.

© 2024 ANSYS, Inc.

Step 4: Converting the pickleball paddle scan data to CAD geometry.

Purpose: Finalize the conversion process from the scan data to obtain the CAD geometry.

1. Navigate to the “Facets” tab and select the “Auto Skin” tool.
2. Select the pickleball paddle scan data and click the “Tick” icon presented on the screen to convert it to a CAD model.
3. Visually check the CAD geometry.

Despite the “Check Facets” tool suggesting that there are no errors within the scan data, when trying to “Auto Skin” it you may be presented with errors like those seen in Figure 8. In which case you should refer to step 3 to complete the extra steps to remove all errors.

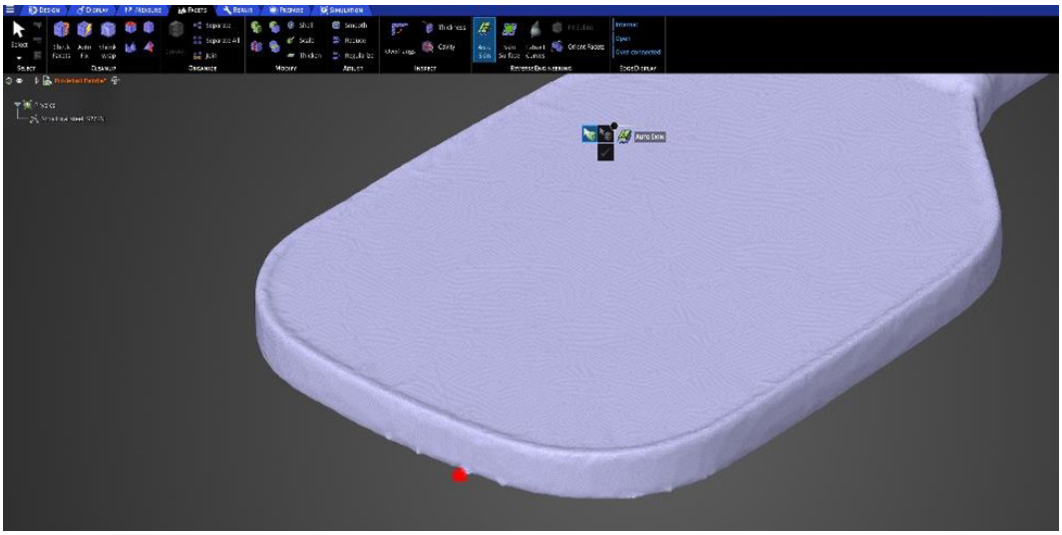

Figure 8: Error when auto skin (Fix sharp tool required)

Once the extra steps have been completed, the scan data should now be prepared for conversion to CAD geometry (Figure 9).

To further enhance the geometry, it is recommended to visually inspect the CAD geometry once converted for any irregularities. As seen in Figure 9(1), the bottom edge of the surface appears uneven.

© 2024 ANSYS, Inc.

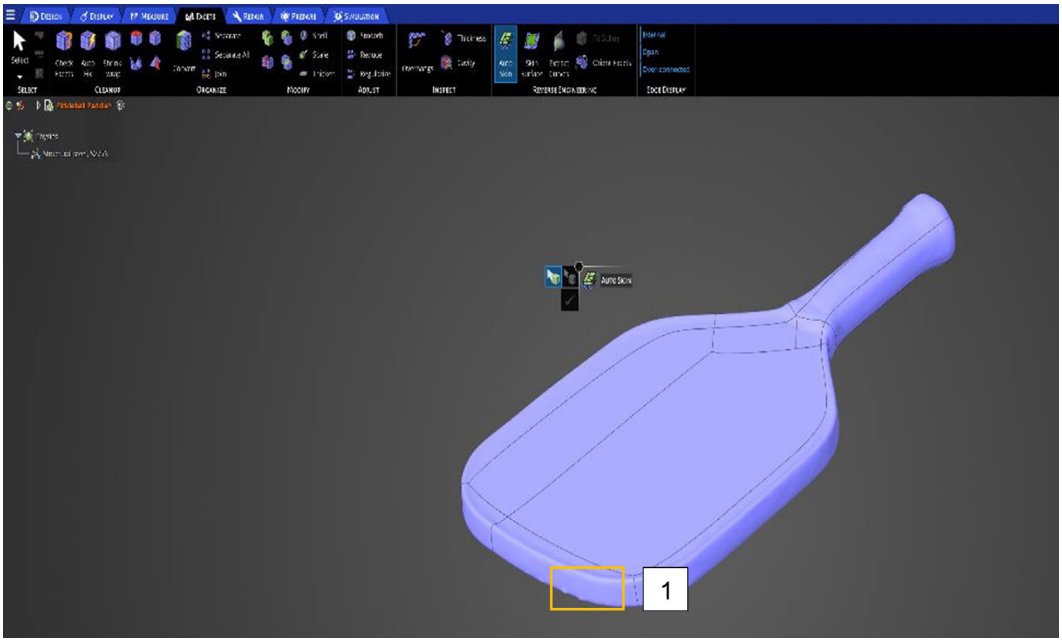

Figure 9: Converted CAD geometry of the pickleball paddle.

Additional smoothing may be beneficial to improve the geometry further using the following tools:

- **Smooth Tool:** Used to smooth out rough facet regions.
- **Reduce Tool:** Used to decrease the number of facets in the faceted body, which can help in refining the geometry.
- **Regularize Tool:** Alternatively, using the Regularize tool can ensure more consistent aspect ratios of selected facets.

Figure 10 shows the final converted CAD geometry.

Figure 10: Converted CAD geometry

© 2024 ANSYS, Inc.

Step 5: Creating a reference frame point for the pickleball paddle.

Purpose: Establishing a reference frame for the pickleball paddle geometry.

1. Navigate to the “Design” section in the ribbon tab.
2. Select the “Move” tool to enable geometry manipulation.
3. Choose the “Select Component” option and select the pickleball paddle CAD geometry.
4. Utilize the triad at the center of the screen for both movement of the geometry and rotation along a specific axis.

Refer to step 5 in Tutorial 1 for more guidance with this step. Figure 11 shows the paddle aligned correctly.

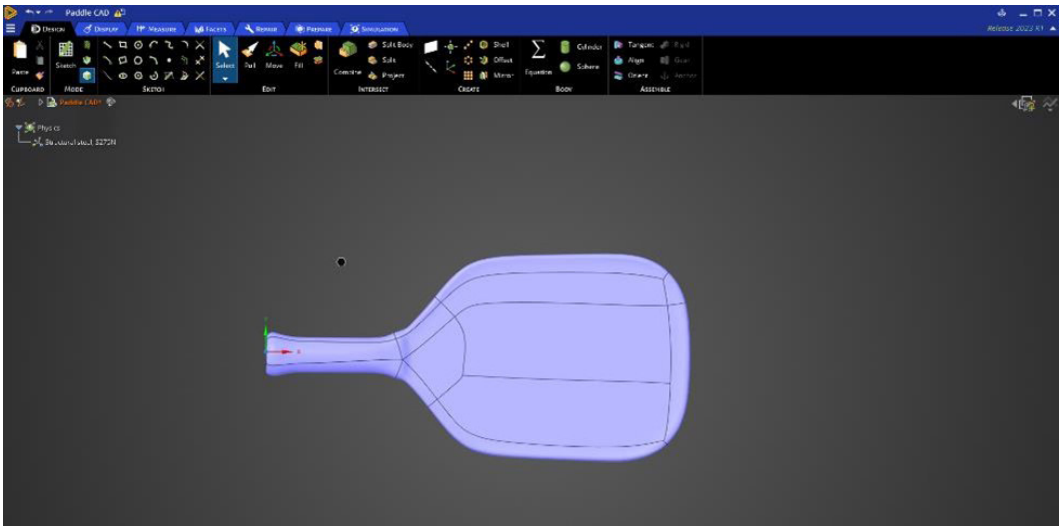

Figure 11: Final geometry displaced to the origin and rotated to align with the y-plane

© 2024 ANSYS, Inc.

Additional Step 6: Applying a mesh to the converted paddle CAD geometry.

Purpose: Apply a mesh to the converted CAD geometry, ensuring it is ready for analysis.

1. Import geometry into the Ansys Mechanical software.
2. Navigate to the “Mesh” section in the Ansys Mechanical toolbar.
3. Select the CAD geometry of the field hockey stick to mesh.
4. Choose appropriate mesh settings for geometry, such as element size and type. In this example, for illustrative purposes, the settings were set to a basic mesh with an element size of “0.005 m” (5 mm).
5. Generate the mesh using “Generate Mesh”.
6. Visually check the mesh to ensure proper coverage and resolution.

Figure 12 shows the paddle with a mesh applied.

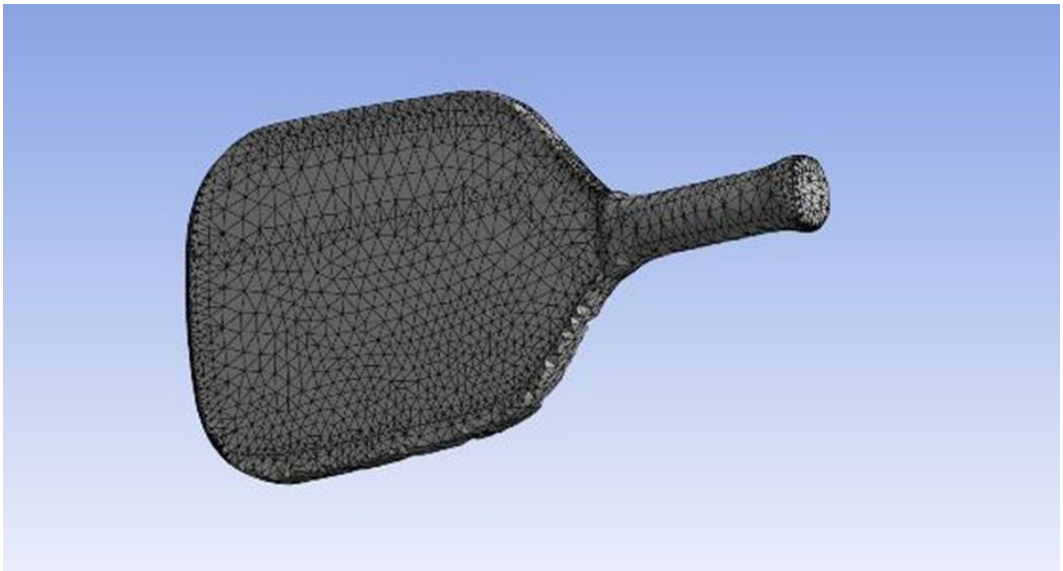

Figure 12: Meshing the pickleball paddle.

© 2024 ANSYS, Inc.

© 2024 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 case study 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.

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

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** 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.

© 2024 ANSYS, Inc. All Rights Reserved.