Manchester Metropolitan University

Creating CAD Geometries in Ansys Discovery Software using 3D Scans

Tutorial 2: Tennis Racket

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

Reviewed by the Ansys Academic Development Team

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 for a tennis racket (shown in Figure 1), clean it up, and convert it to CAD geometry. We will be following similar steps to the field hockey stick tutorial (Tutorial 1 in this series). Less guidance is provided in this example.

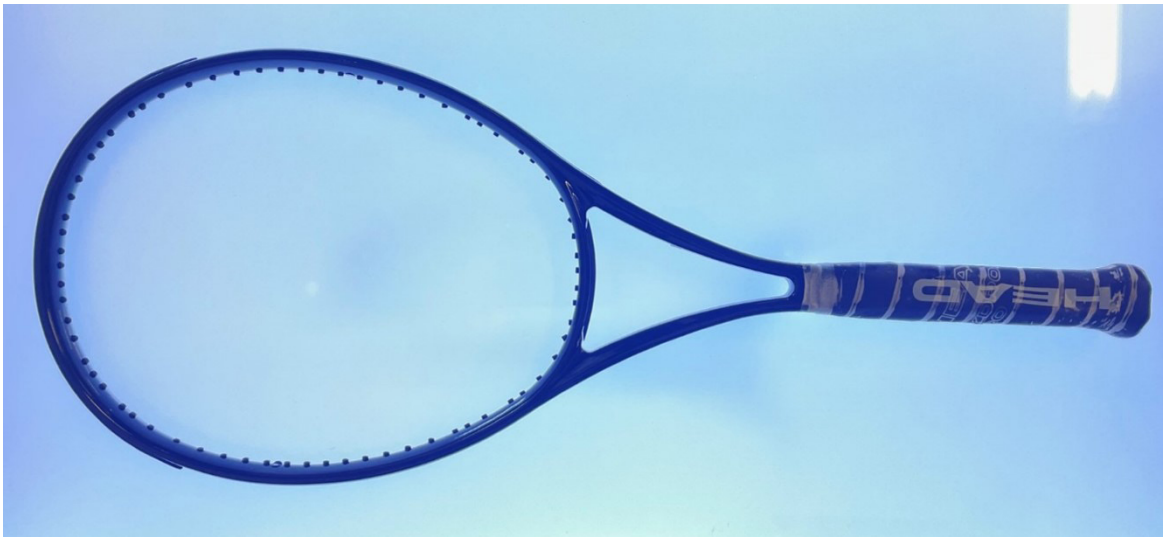

Figure 1: Tennis racket scanned for this tutorial

Table of Contents

Step 1 Importing the tennis racket STL file

Purpose: Import the STL file generated from the 3D scan of the tennis racket into the Ansys Discovery software, preparing its conversion to CAD geometry.

1. Launch the Ansys Workbench software.
2. Navigate to the geometry, right-click, and select “Import Geometry” → “Browse”.
3. Select the provided “Tennis racket (NO holes)” STL file and click “Open”.
4. Alternatively, open Ansys Discovery software,click the “File Menu” button, and select “Open”.
5. Select the “Tennis racket (NO holes)” STL file.

The imported STL geometry can be seen in Figure 2.

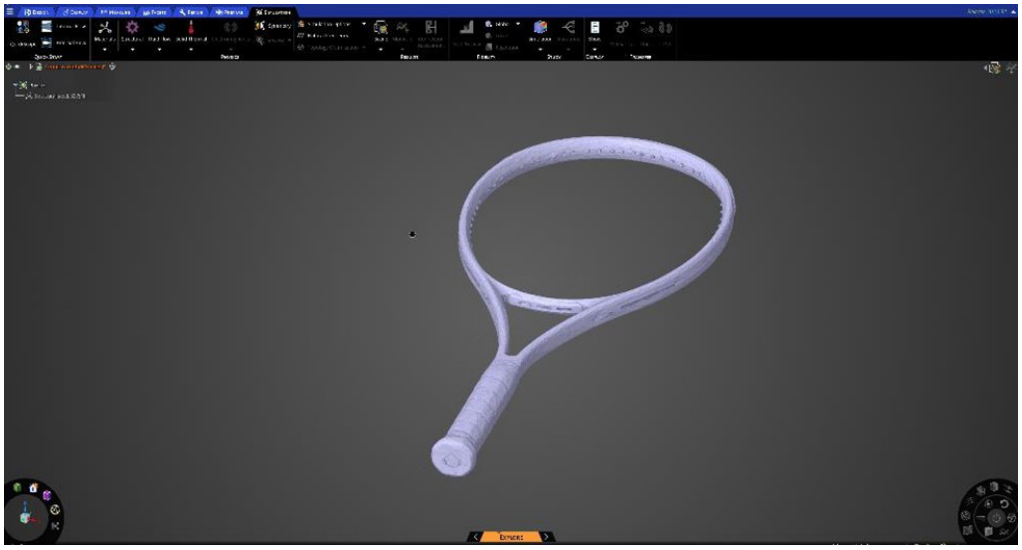

Figure 2: STL file of the tennis racket scan data in the Ansys Discovery software.

Step 2: Checking for errors in the tennis racket 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, like the process outlined in Tutorial 1.

The scan data errors are shown in Figure 3.

3. Visualize any errors using the “Holes”, “Intersections”, “Over-Connections, and “Fix Sharps” tool. Use “Auto Fix” to automatically fix errors within the geometry.

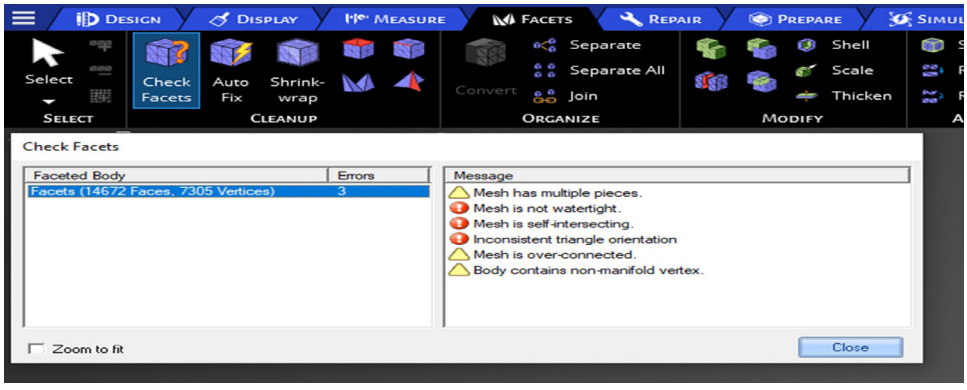

Figure 3: Tennis racket scan data errors

For more details regarding this step, refer to Part 1 Step 2 within Tutorial 1.

Step 3: Cleaning up the tennis racket scan data.

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

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

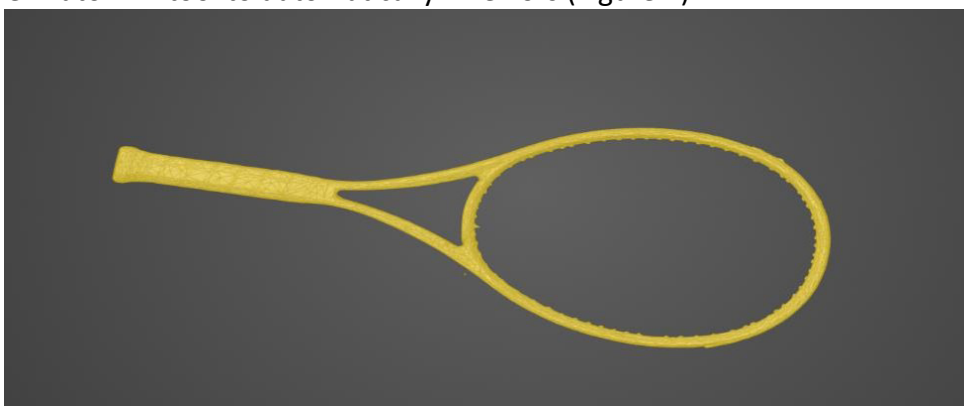

Figure 4: Tennis racket scan data after applying the “Auto Fix” tool

2. Select “Check Facets” to check any remaining errors within the scan data.
3. Further refine the data using “Shrink wrap”, to close any remaining gaps or imperfections. Entering a size of “1 mm” within the size tab (Figure 5).

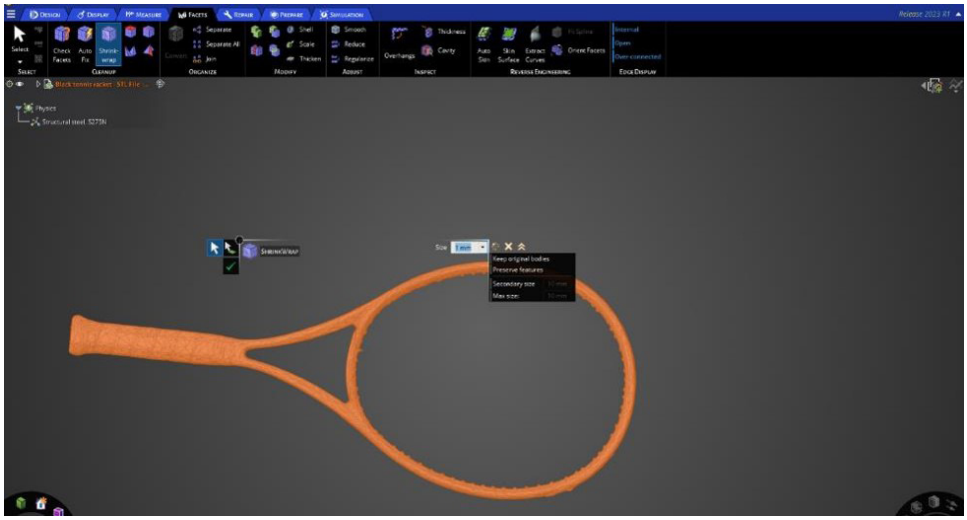

Figure 5: Shrink-wrapping the scan data with 1mm

4. Use the “Fix Sharps”, “Over-Connected”, “Holes”, and “Intersections” cleanup tools to identify and manually fix any remaining errors if necessary (Figure 6).

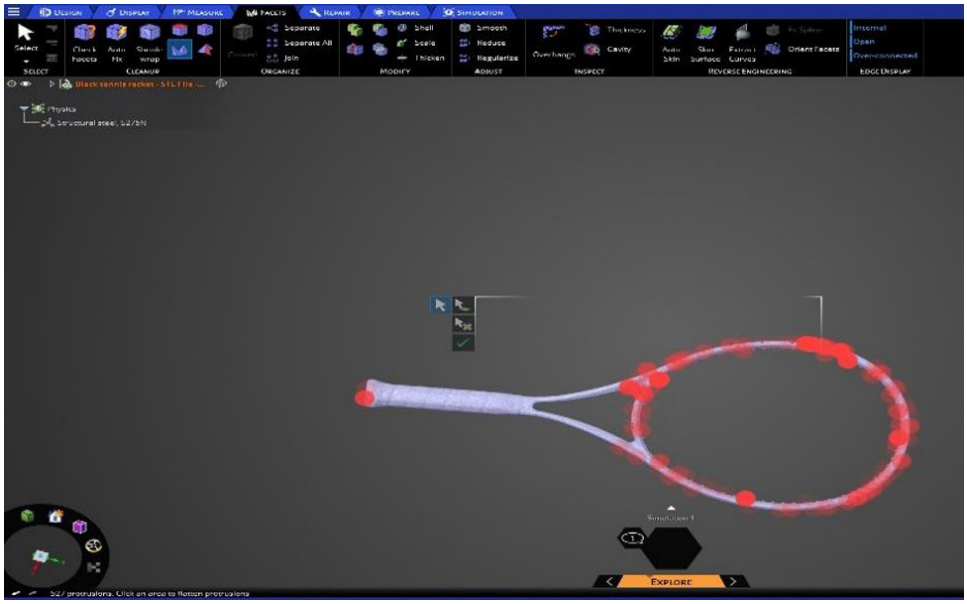

Figure 6: “Fix Sharps” errors within scan data

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

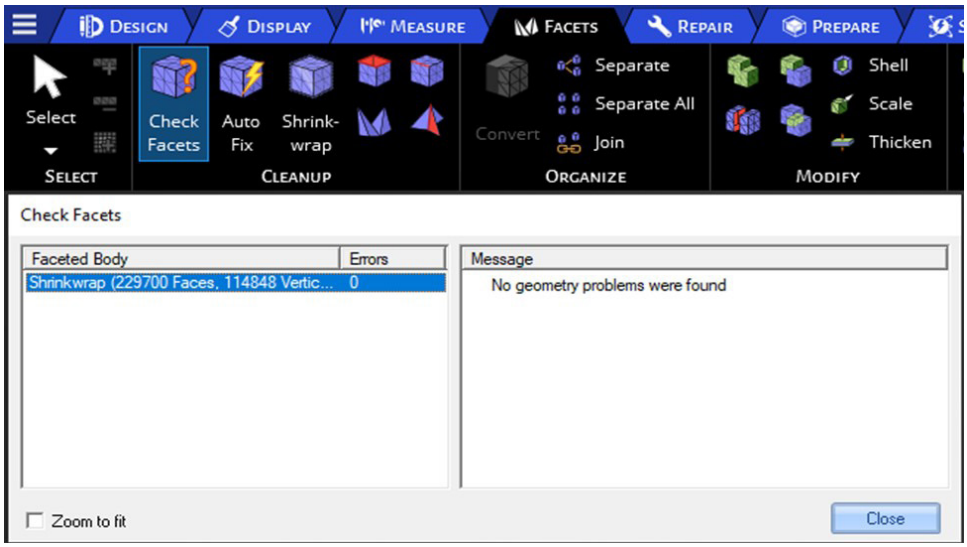

Figure 7: Message showing there are no errors within the scan data after all points in step 3 have been completed.

Step 4: Converting the tennis racket scan data to CAD geometry.

Purpose: Finalize the conversion process from the scan data to obtain the CAD geometry, following similar steps outlined in Section 1 – Step 4 of the tutorial.

1. Navigate to the “Facets” ribbon tab, as outlined in Step 2
2. Select the “Auto Skin” tool, shown in Figure 8.
3. Choose the tennis racket scan data and select the tick presented on the screen to convert the scan data to CAD geometry, as shown in Figure 9.

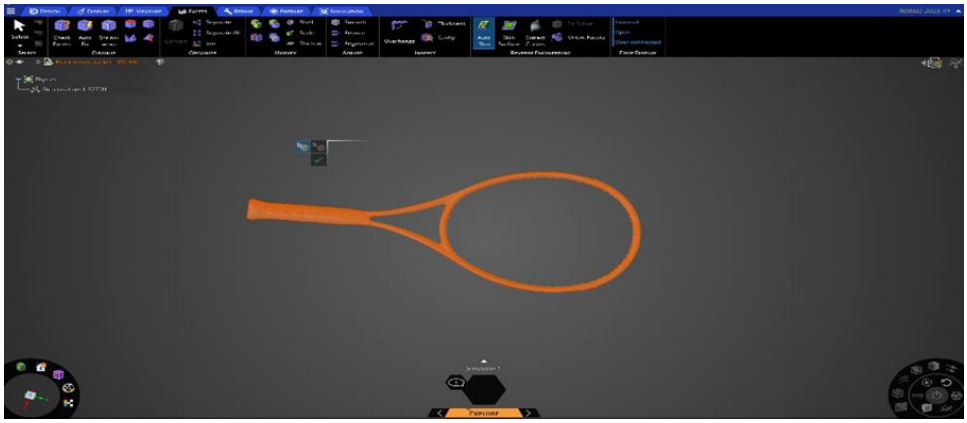

Figure 8: Converting the tennis racket scan data to CAD geometry using the “Auto Skin” tool.

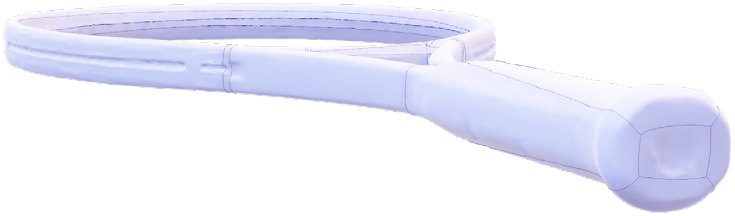

Figure 9: Tennis racket CAD geometry

Step 5: Creating a reference frame point for the tennis racket.

Purpose: Establishing a reference frame for the tennis racket geometry, for the purpose of future analysis.

1. Navigate to the “Design” section in the Ansys Discovery software (Fig. 10).

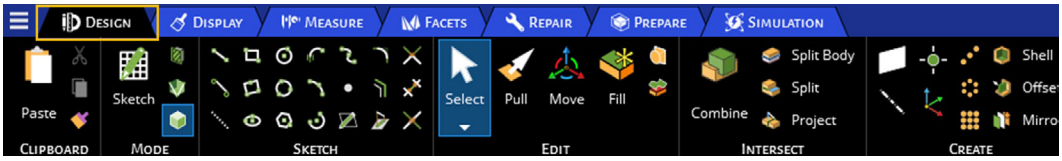

Figure 10: Selecting Design tab.

2. Select the “Move” tab (Fig. 11).

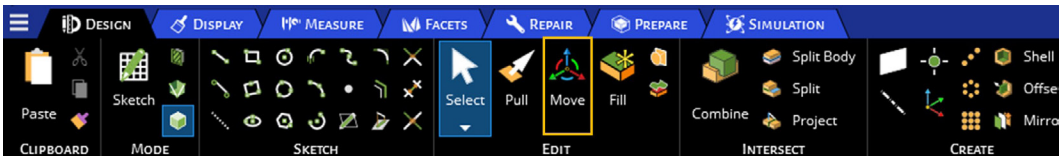

Figure 11: Selecting Move tool.

3. Select the “Select component” option (Fig. 12 (1)) and select the tennis racket.

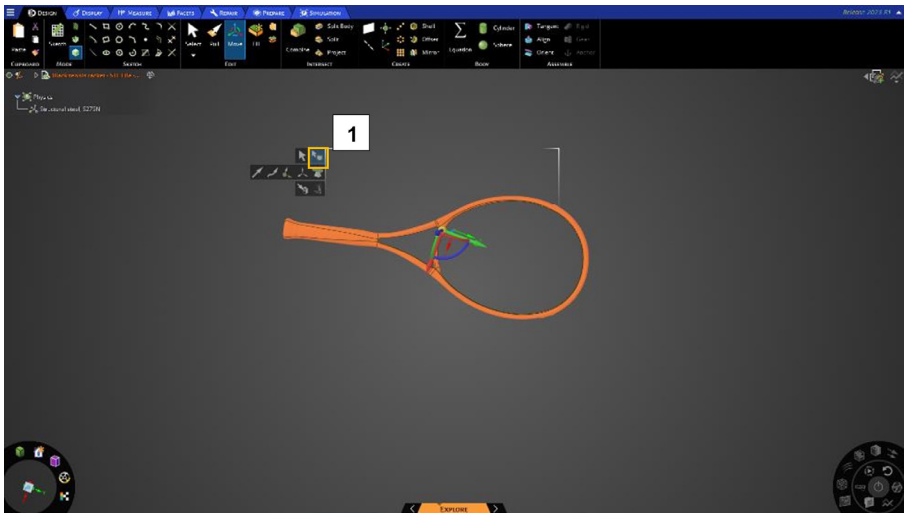

Figure 12: Image showing the “Move” command options.

4. Select the arrow corresponding to the desired direction, align the racket with the origin, on the triad and drag it with your mouse. Use the dialog box if you wish to specify a specific displacement value (Fig. 13 (1)).

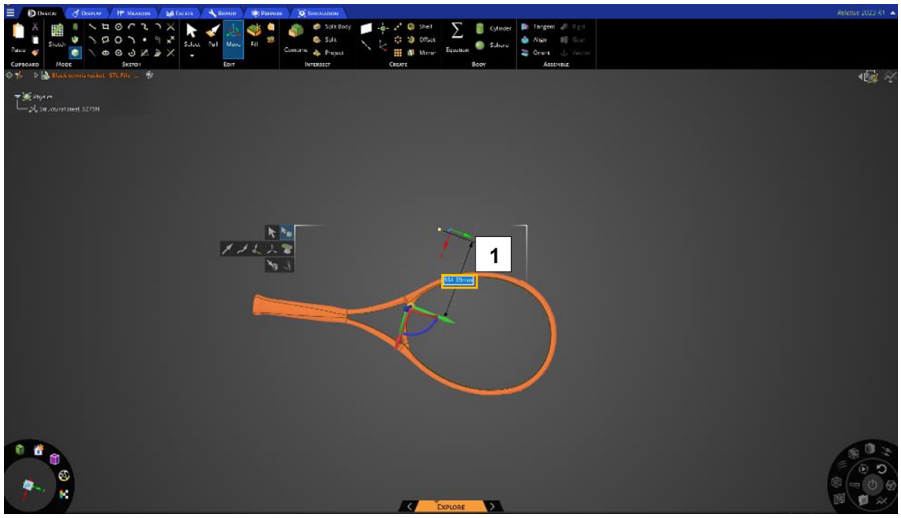

Figure 13: Displacing the geometry.

5. Choose the desired axis of rotation on the triad and drag with your mouse. If a precise angle of rotation is required, enter it into the dialogue box (Fig. 14 (1)). The final CAD geometry aligned with the origin and y-axis can be seen in Figure 15.

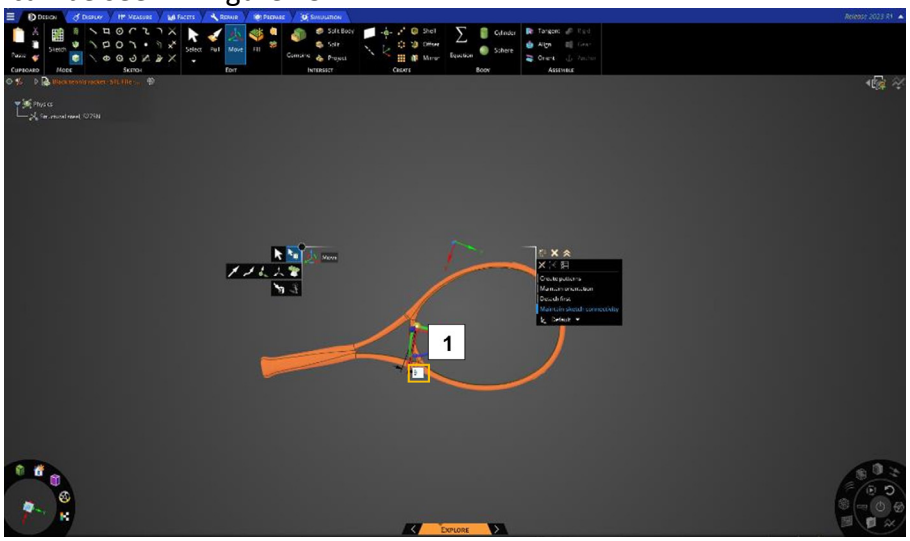

Figure 14: Rotating the geometry.

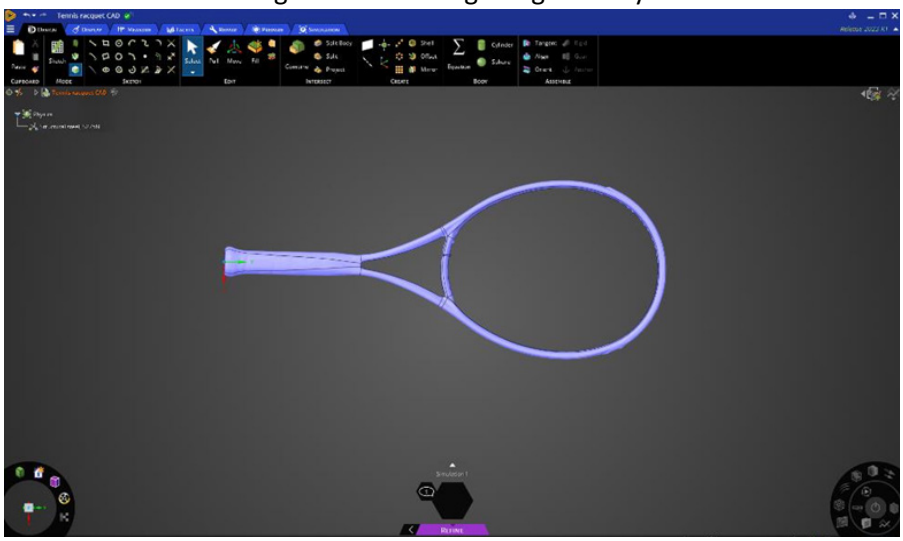

Figure 15: CAD geometry of the tennis racket displaced to the origin and rotated to align with the y-axis.

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

Purpose: Prepare the CAD geometry for FEA by applying a mesh.

Now the CAD geometry has been created and aligned within the global coordinated system, we are ready to mesh it in preparation for FEA. This step shows the ability to mesh the CAD geometry for further analysis. Users are again encouraged to explore additional tutorials that focus on meshing techniques, considering their available computational resources.

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 review the mesh to ensure proper coverage and resolution.

The example mesh can be seen in Figure 16.

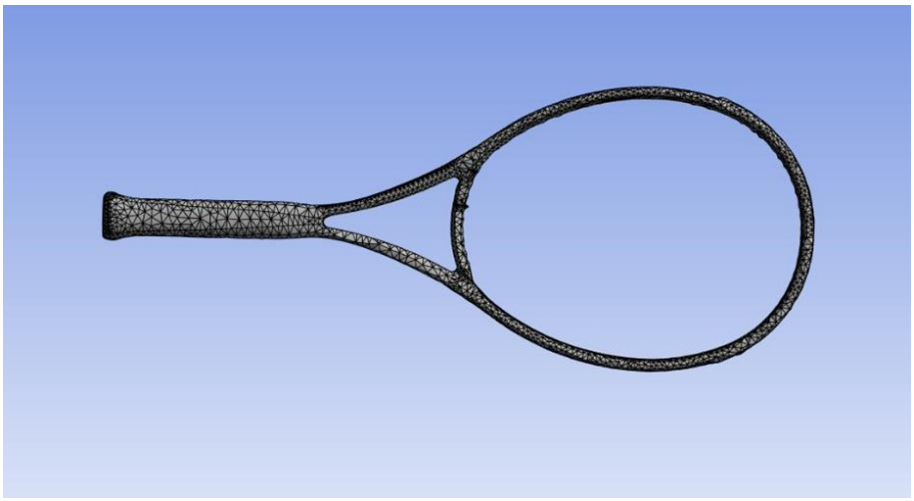

Figure 16: Mesh applied to the tennis racket CAD geometry.

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