Creating CAD Geometries in Ansys Discovery Software using 3D Scans

Tutorial 1: Field Hockey Stick

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 the field hockey stick in shown in Figure 1 and convert it into a format that can be edited in the CAD environment, while identifying and cleaning any imperfections.

Figure 1: Field hockey stick scanned for this tutorial

Table of Contents

Step 1: Importing the field hockey stick stereolithography file

Purpose: The first step is to import the stereolithography (STL) file generated from the 3D scan of the hockey stick into the Ansys Discovery software, providing the data for conversion to CAD geometry.

1. Launch the Ansys Workbench software.
2. Navigate to the geometry (Fig. 2 (1)), right click and select “Import Geometry” (2) – “Browse” (3).
3. Choose the scan STL file of the hockey stick which has been provided in the supporting files “Field Hockey Stick STL”, click “Open.”
4. You are now ready to “Open” “New Discovery Geometry” to see the scan file in the Ansys Discovery Software.

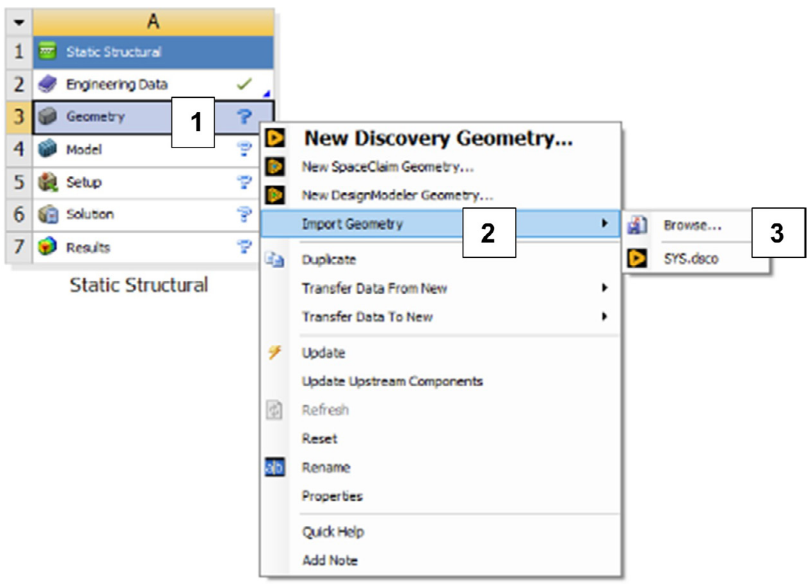

Figure 2: Steps to import the geometry using the Ansys Workbench tool

Alternatively:

1. Launch the Ansys Discovery software.
2. Click the three lines in the top left of the screen, seen in Figure 3, to open the “File Menu” (1), select “Open” (2) to open your files.
3. Choose the scan STL file of the hockey stick which has been provided in the supporting files “Field Hockey Stick STL”, click “Open.”

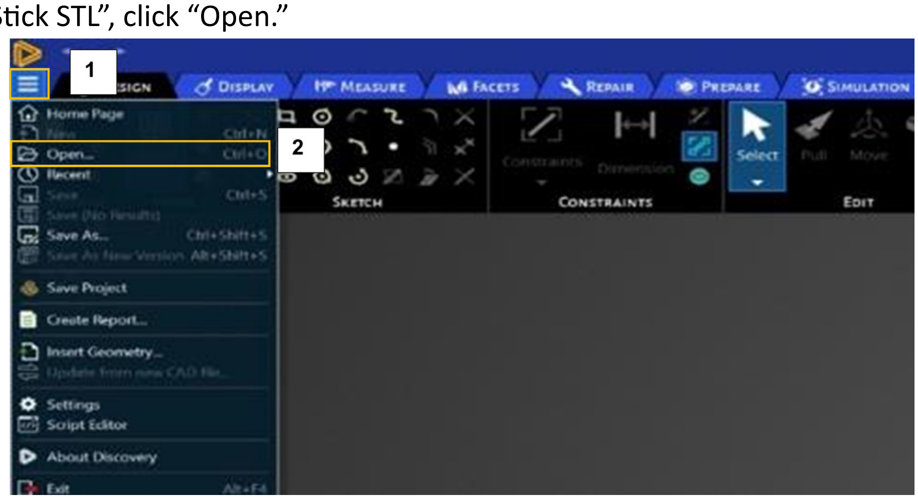

Figure 3: Steps to import geometry using the Ansys Discovery Software

Figure 4 shows the imported STL geometry in the Ansys Discovery interface.

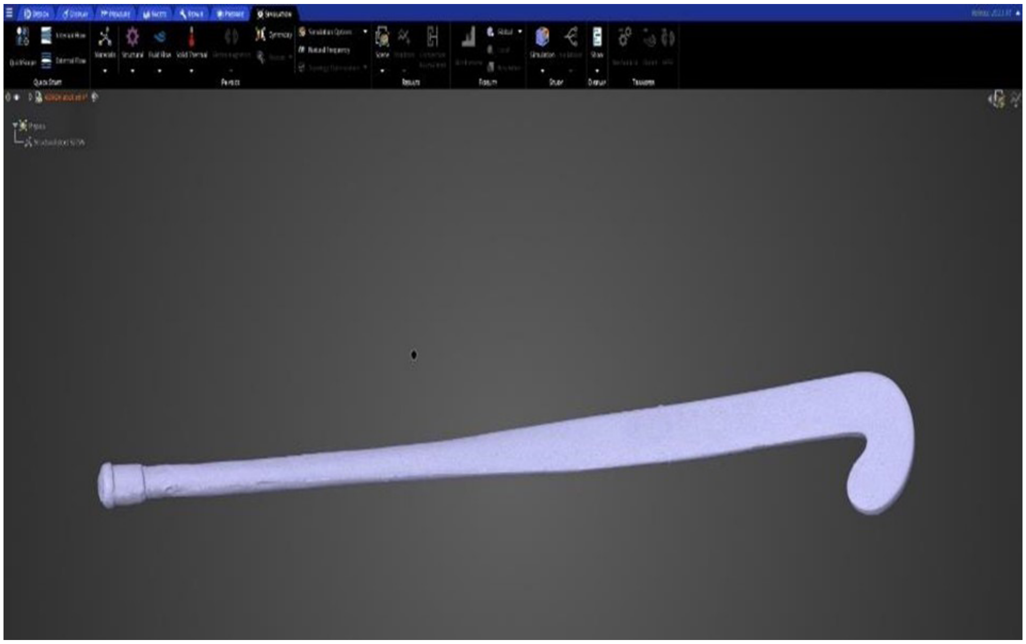

Figure 4: STL file of the uploaded field hockey stick

Step 2: Checking for errors in the field hockey stick scan data

Purpose: This step involves identifying any errors and imperfections within the scan data, which would prevent us from converting it to a CAD geometry.

1. After importing the STL file, go to "Facets" in the "Ribbon Tab" highlighted in Figure 5, to display the facet tools (1).

Figure 5: (1) Selecting Facets in the Ribbon Tool

2. Select "Check Facets" tool (Fig. 6) to check for any errors in the scan data (2), listing any problems with the faceted body.

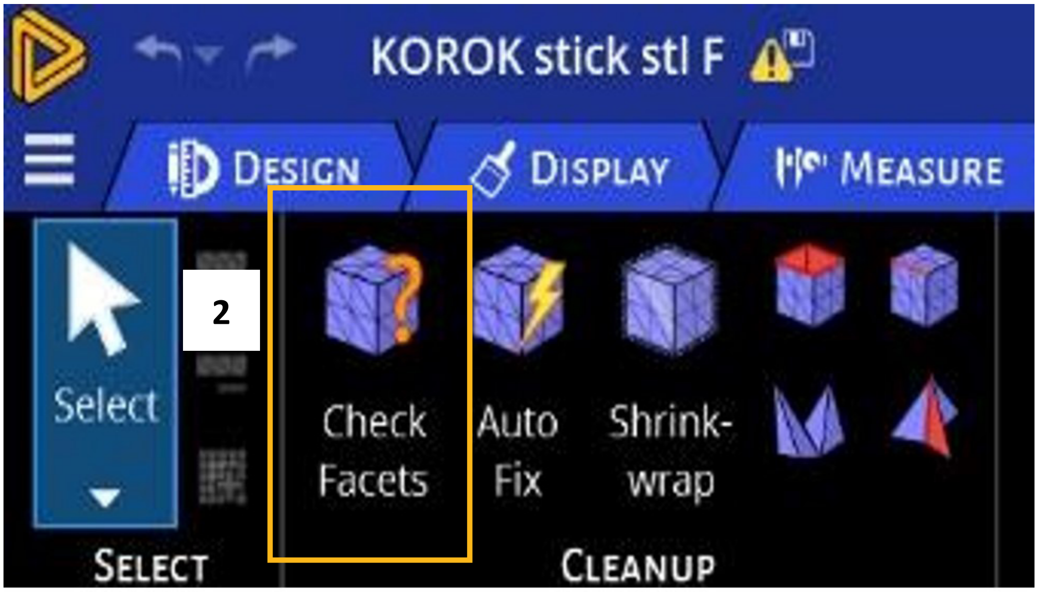

Figure 6: (2) Tool used to check for errors (Check Facets)

The errors in the scan geometry should appear in the following menu format seen in Figure 7. Such errors mean the CAD geometry cannot be created.

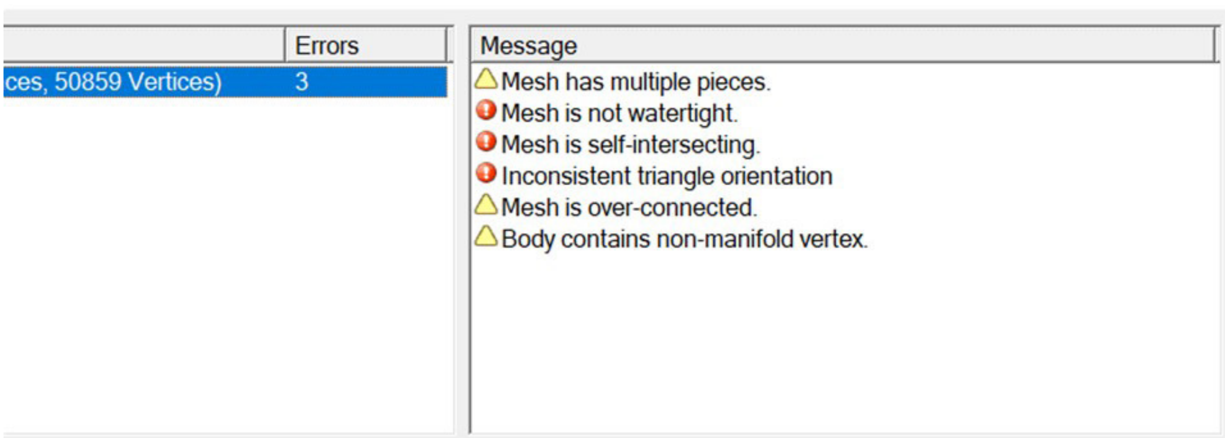

Figure 7: Image showing errors identified in the scan geometry using the "Check Facets" function.

3. To visualize the errors, select between the "Intersections", "Fix Sharps", "Over-Connected" and "Holes" tools. These tools can also be used to fix individual errors in the scan data.

Where:

- "Intersections" tool identifies the areas of self-intersection within the data.
- "Fix sharp" tool can be used to identify and fix any sharps points within the data.
- "Over-Connected" tool can be used to find and fix facets connected to other facets.
- "Holes" can be used to identify and fix holes within the faceted body.

Step 3: Cleaning up the field hockey stick scan data.

Purpose: In this step, we will address any errors or imperfections identified in the scan data to prepare it for conversion to CAD geometry.

Figure 8: Tool to reduce these errors in scan geometry (Auto Fix)

1. To automatically fix errors within the geometry, select "Auto Fix" (Fig. 8 (1)). The geometry options for Auto Fix can be seen in Figure 9.
2. Once this has completed, click "Check Facets", as instructed in Step 2, to view any remaining errors.

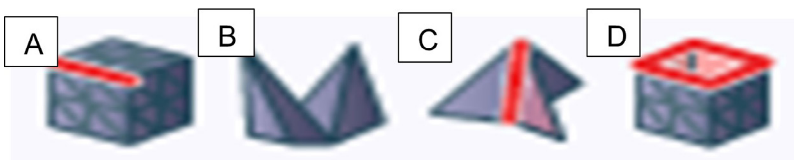

Figure 9: Various geometrics for Auto Fix
(A) Intersections, (B) Fix Sharps, (C) Over-Connected, (D) Holes

3. If there are remaining errors, you can manually fix them using the tools highlighted in the yellow square (Fig. 10 (1)). This highlights the errors within the scan data, as can be seen in Figure 10 indicated by the red spots, showcasing the "Fix Sharp" errors when the tool is selected. This can be repeated for the "Intersections", "Overconnected", and "Holes" tools as mentioned in the previous step.

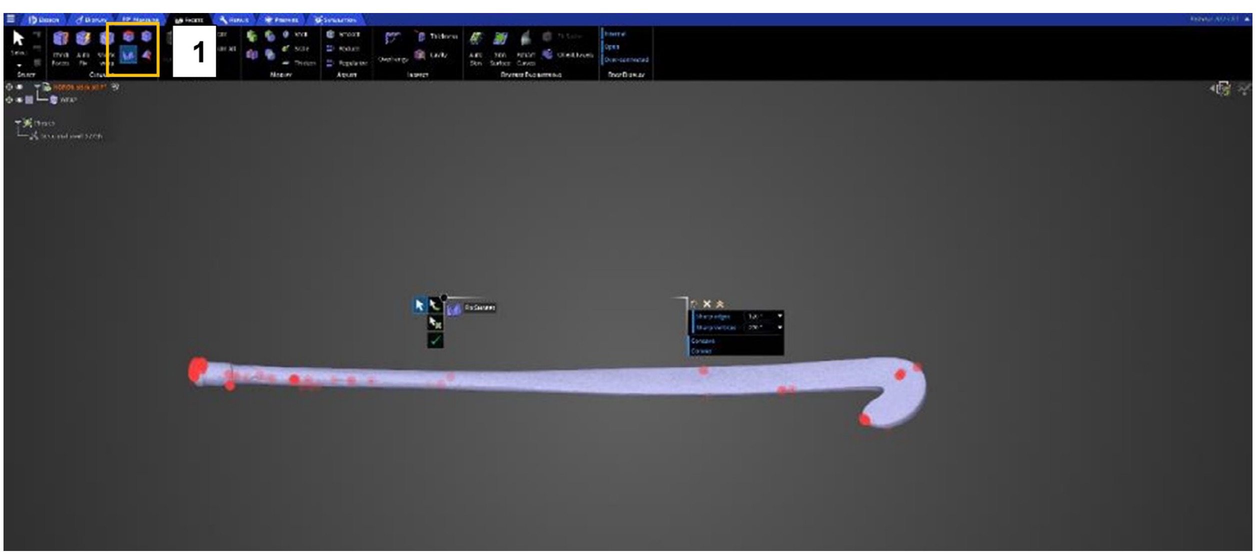

Figure 10: Manual fixing of errors in scan data

4. To further refine the scan data, select the “Shrink Wrap” tool (Fig. 11 (1)), which creates a single faceted wrap around the geometry to close any gaps.

Figure 11: Selecting the Shrink Wrap

5. Once “Shrink Wrap” has been selected, using the drop-down tab select the “Hockey Stick STL” geometry (Fig. 12 (1)), which then allows you to adjust the size of the shrink wrap (Fig 12-(2)). Enter “1 mm” in the size tab.

**While a default size of “1 mm” is suggested for this example, this varies depending on the quality of the scan geometry. Varying the size impacts how well the imperfections of the geometry are smoothed out and can impact the ability to convert it to CAD geometry. It is recommended to explore different settings to showcase the potential of this tool.

6. Then finally, select the “Tick” (Fig 12-(3)), to apply the shrink wrap to the geometry.

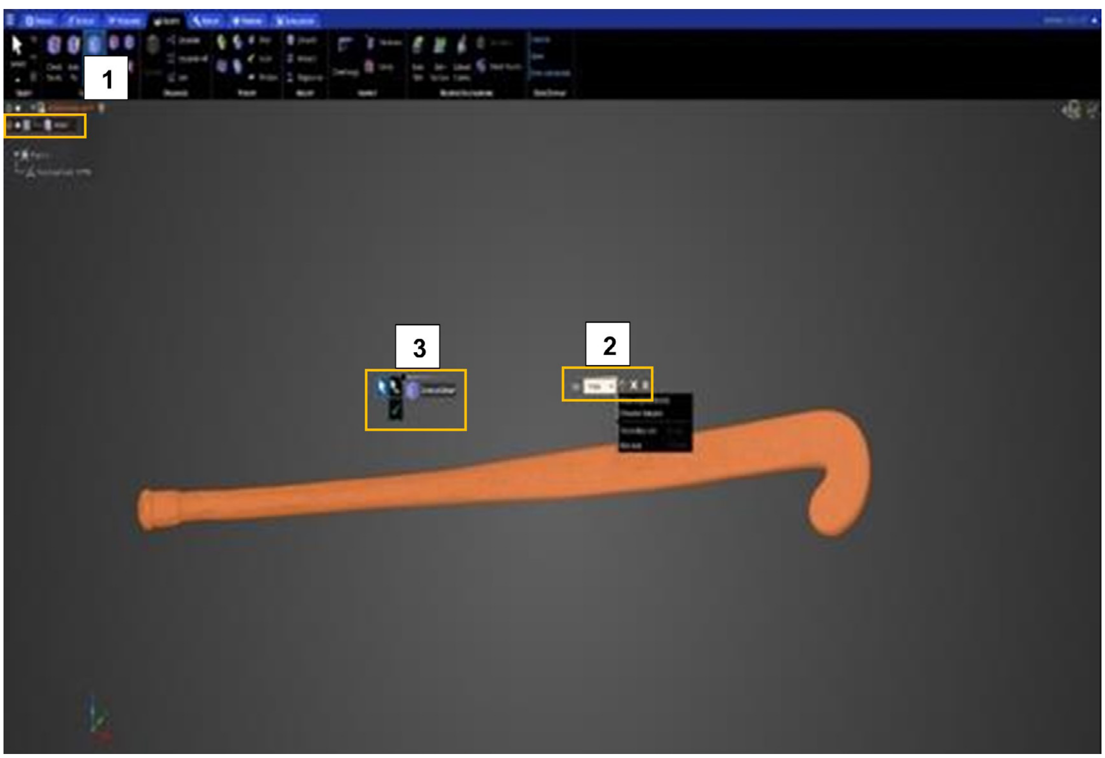

Figure 12: Shrink wrapping the scan data

To check if the faceted body is ready to be converted into CAD geometry, select the “Check Facets” tab as instructed in Step 2. You should then be presented with the message confirmation presented in Figure 13. This message confirms that the scan data has “No geometry problems”, suggesting it is suitable for the next steps.

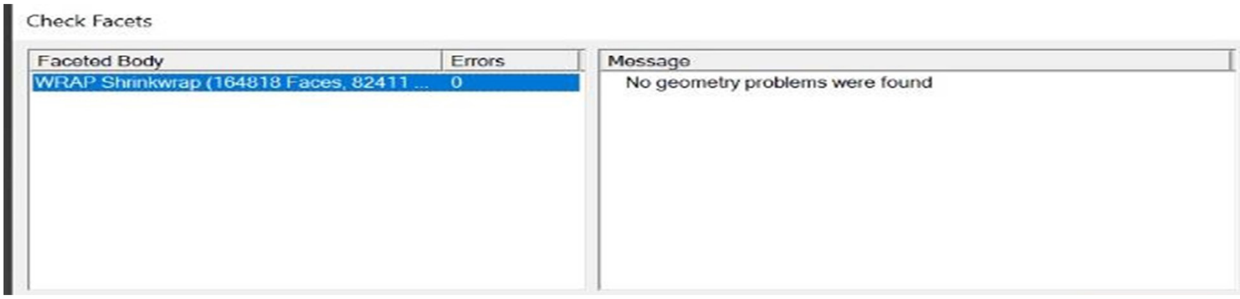

Figure 13: Checking Facets after Shrink Wrap

Step 4: Converting field hockey stick scan data to CAD geometry.

Purpose: In this step, we will convert the hockey stick scan data into CAD geometry ready for analysis.

1. Begin by selecting “Facets” in the ribbon tab, then locate “Auto Skin” (Fig. 14).

Figure 14: Finding the Auto Skin tool

2. Select the scan data (Fig. 15 (1)), and then select the “Tick” (2) to convert into CAD geometry, as seen in Figure 15.

Figure 15: Image showcasing the next steps to complete the Auto Skin on the field hockey stick

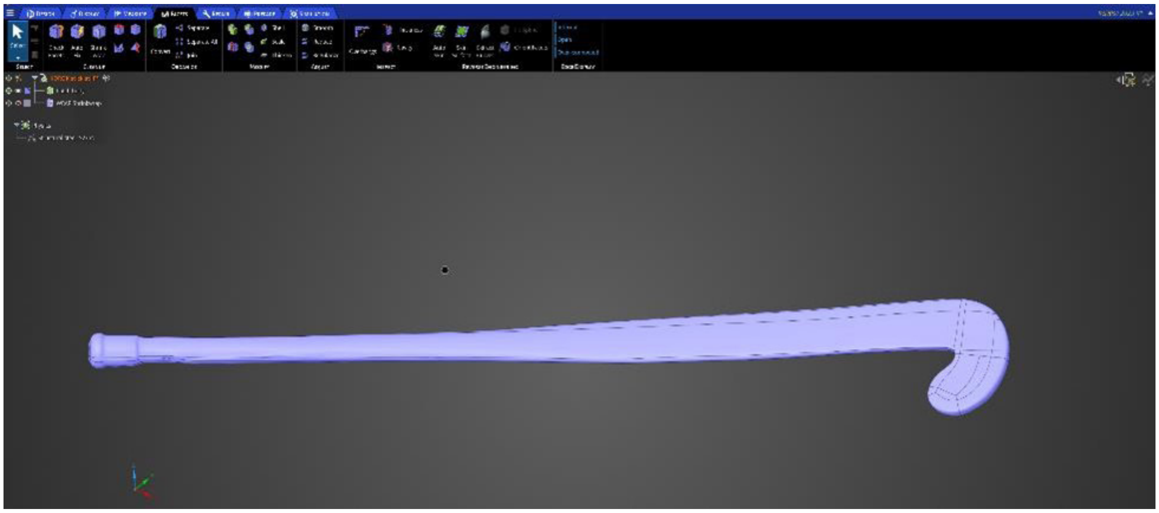

Figure 16: CAD geometry of the field hockey stick obtained from the scan data.

Step 5: Creating reference frame point for the field hockey stick.

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

1. Navigate to the "Design" section in the Ansys Discovery toolbar (Fig. 17).

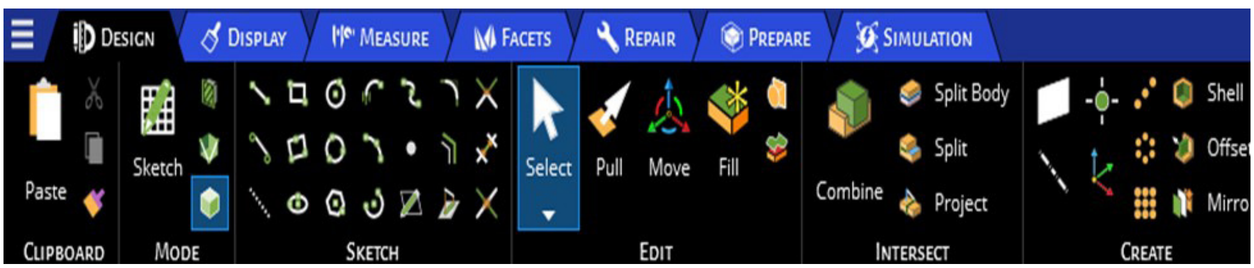

Figure 17: Selecting Design tab.

2. Select the "Move" tab (Fig. 18).

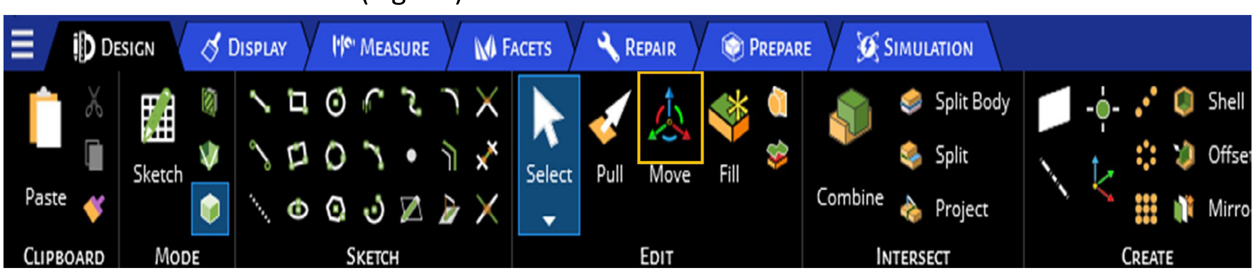

Figure 18: Selecting Move tool.

3. Select the "Select component" option (Fig. 19 (1)), and select the hockey stick, to enable it to be displaced. The triad in the center of the display (Fig. 19 (2)) is used to move and rotate the geometry relative to an axis.

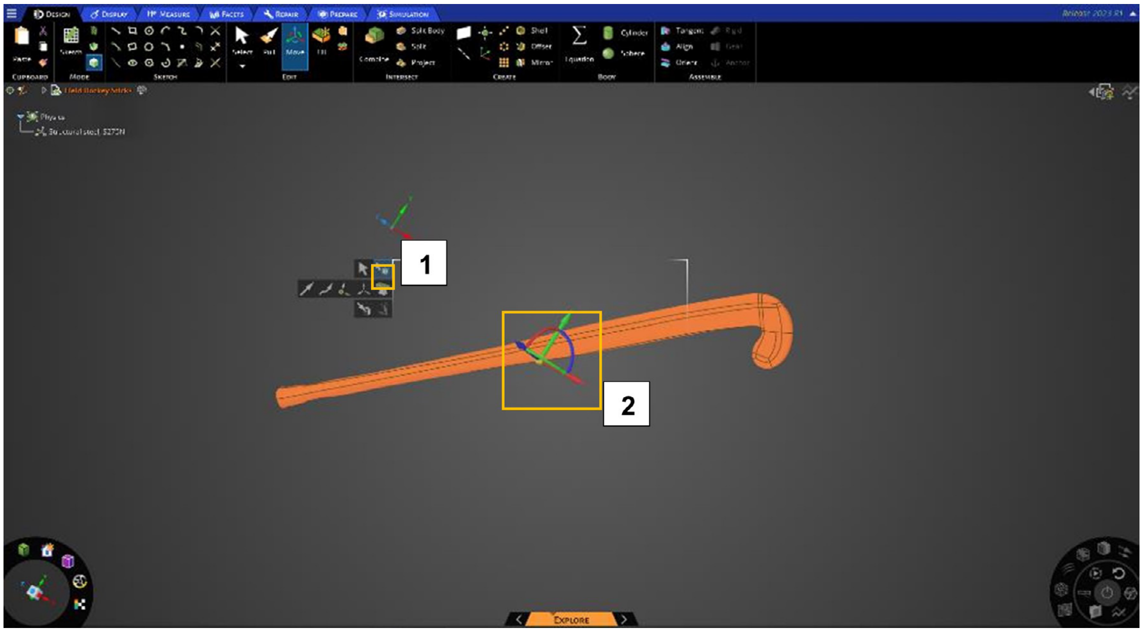

Figure 19: Image showing the move command options.

The triad's green, red, and blue arrows represent the y-, x-, and z-axis, respectively. To move the geometry along an axis, select the corresponding arrow on the triad (Fig. 20) and drag it with your mouse. If you need to specify a precise displacement, enter the value in the dialog box shown in (Fig. 20 (1)).

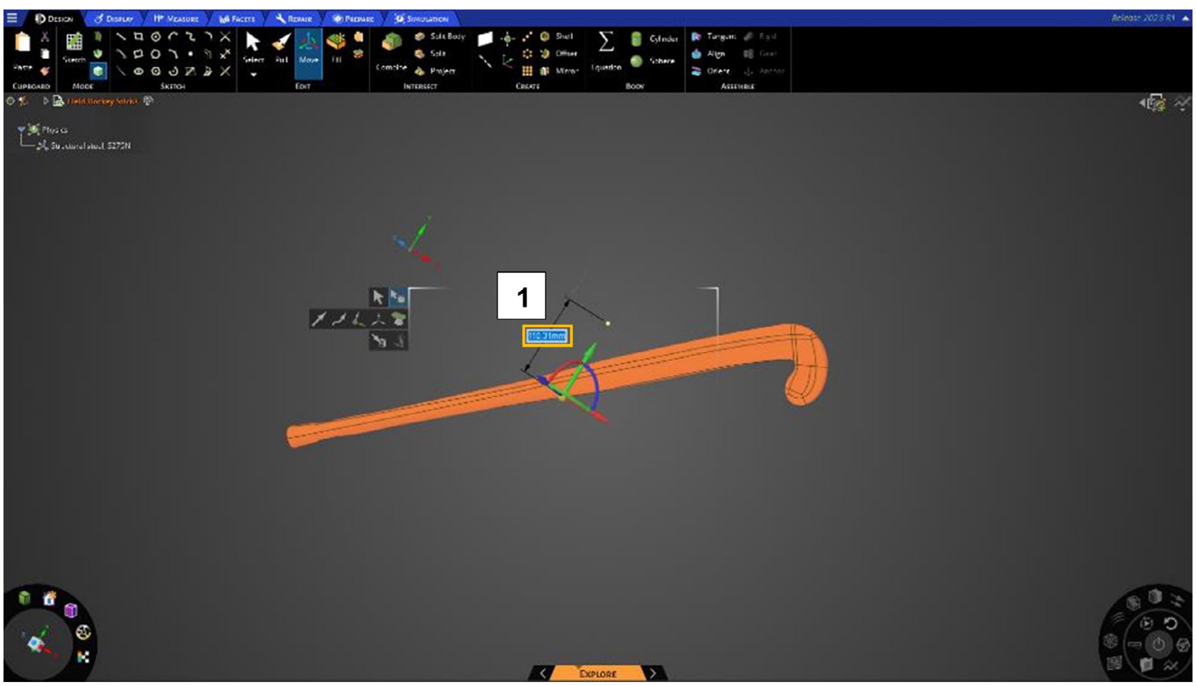

Figure 20: Displacing the converted field hockey stick CAD geometry.

Rotation of the geometry is facilitated by the green, red, and blue arcs on the triad, representing rotation around the y-, x-, and z-axis, respectively. To rotate the geometry, select the desired axis of rotation on the triad (Fig. 21) and drag with your mouse. For precise rotation angles, enter the values in the dialog box. The geometry aligned with the origin and the y-axis can be seen in Figure 22. The final converted CAD can be seen in Figure 23.

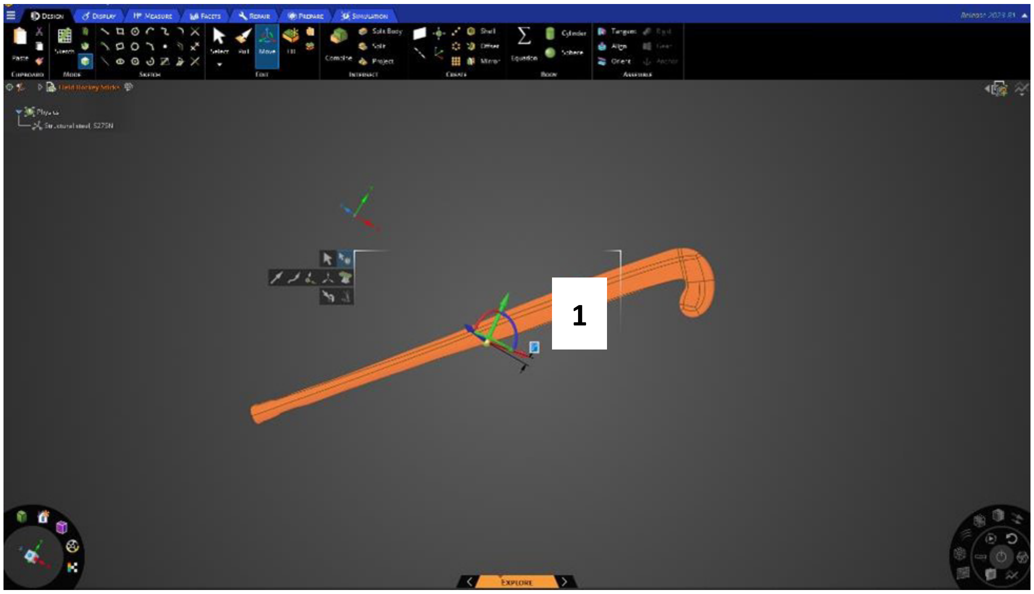

Figure 21: Rotating the geometry.

Figure 22: CAD geometry of the hockey stick displaced to the origin and rotated to align with the y-plane.

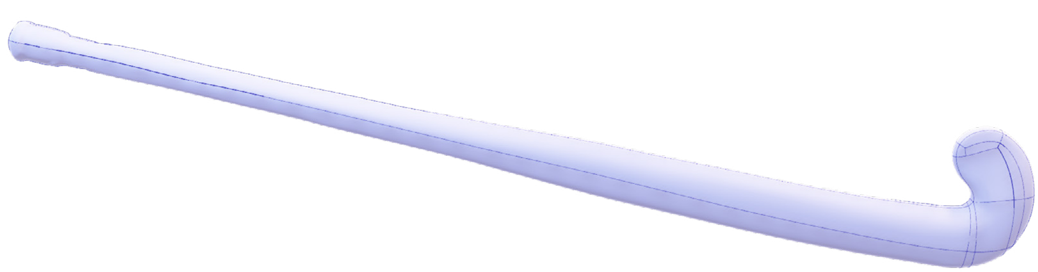

Figure 23: Converted field hockey stick CAD 3D model

Optional Step 6: Applying a mesh to the converted field hockey stick 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 apply mesh, preparing it for FEA. This step shows the ability to mesh the CAD geometry for future analysis. Users are encouraged to explore additional tutorials that focus on meshing techniques, considering computational resources.

1. Import geometry into the Ansys Mechanical software.
2. Navigate to the "Mesh" section in the Ansys Mechanical menu.
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 24.

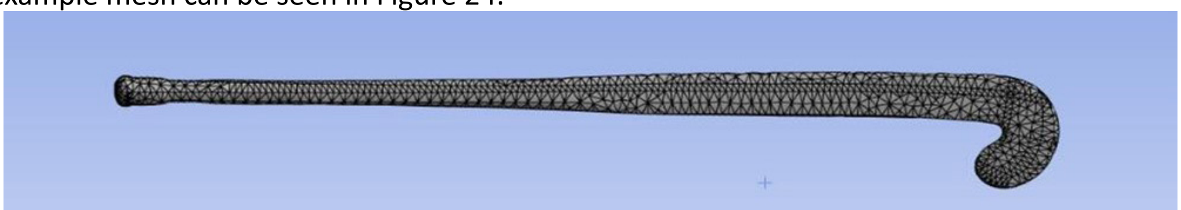

Figure 24: Field hockey stick meshed geometry

Further Work: Exploring Finite Element Analysis.

While this tutorial has focuses on the conversion of the scan data to CAD geometry, Ansys also offers learning resources on FEA using the Ansys Mechanical software. These resources can provide further guidance in the development of finite element models using the CAD geometries produced in this tutorial. These resources offer guidance on setting up simulations, defining material properties, applying boundary conditions, and gathering desired results. To learn more about FEA and its applications, learners are encouraged to explore the tutorials and resources available on the Ansys educational resources site, such as the Basics of FEA in Ansys Mechanical Tutorial. An example FEA simulation for the hockey stick can be seen in Figure 25.

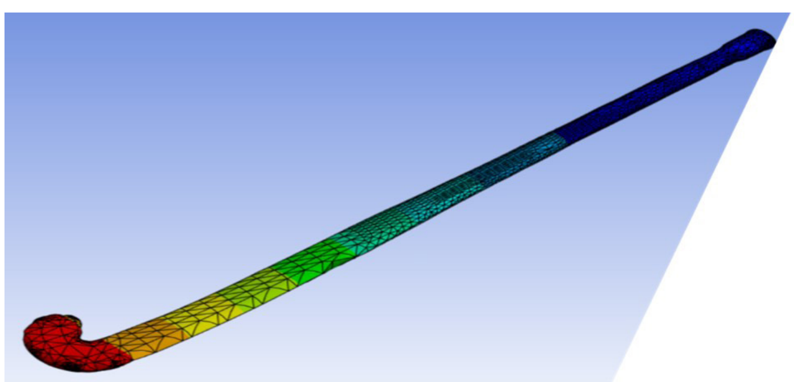

Figure 25: FEA example on the field hockey stick CAD geometry with a fixed support at the grip and a force applied at its free end.

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