READ ME

Getting Started with PyMAPDL

Developed and curated by the Ansys Academic Development Team

Sahil Sura

education@ansys.com

Ansys Software Used

This resource uses Ansys Mechanical™ structural finite element analysis software via the Python-based interface PyMAPDL.

1. Overview

This exercise helps users get started with PyMAPDL, a Python based interface for interacting with the APDL solver with a hands-on tensile test simulation based on the ASTM E8 standard. The example demonstrates how Python can be used to script, automate, and visualize finite element simulations, making advanced engineering analysis more accessible and interactive. By integrating PyMAPDL with Python's scientific libraries, students gain exposure to modern simulation workflows.

2. Learning Objectives

Users will be able to:

1. Create a parametric geometry using scripting
2. Set up a static structural simulation with PyMAPDL
3. Extract results using user defined commands
4. Understand the simulation setup and PyMAPDL interaction

3. Student Knowledge Prerequisites

It is expected that the user has a basic understanding of Python scripting and a general awareness of material properties and testing concepts. No prior experience with Ansys or APDL is required. This case study explores how simulations work and is mainly targeted toward curious learners who are open to learning by doing, including those who are new to engineering simulation tools.

4. PyAnsys and Scripting with Ansys

This resource utilizes the open source PyMAPDL library, developed by Ansys as part of the PyAnsys initiative to allow Ansys simulation tools to be controlled via programming through Python. This allows the power of simulation and visualization to be presented to students in a simpler interface for introductory courses, such as here where a simulation can be run, and displacement and stress fields plotted, from inside a Jupyter Lab environment – without needing to display the Ansys Mechanical software/ MAPDL user interface.

5. System/ Library Requirements

In order to run, the following software and libraries must be installed on the computer. (The version numbers listed below in brackets are those which were used to create this resource; while it may run successfully with other versions, this has not been tested.)

1. Python (3.10.11)
2. Jupyter Lab (4.4.9)
3. PyMAPDL Libraries:
   » Core (0.71.0)
   » Visualization
   » Core [Graphics]

4. Additional Python Libraries
   » Numpy
   » Pyvista (0.45.3)

If you are unfamiliar with installing Python packages, guidance can be found on the Python website or elsewhere online.

6. Prerequisites Jupyter Lab First Time Setup

1. Open the zip file and extract its contents to a single location. The virtual lab is contained within the Jupyter Notebook file "Getting_started_with_PyMAPDL.ipynb".
2. Before demonstrating it to students, it is suggested running the simulation to understand how long it will take on the computers where you have it installed, especially if you wish to take advantage of the option not to show them Ansys Mechanical software/APDL user interface. To do this:
   » Use the "Run all" button in Jupyter Lab to run all cells.
   » APDL icon is visible on the taskbar ensuring the APDL solver is being accessed for the simulation.
   » You now know how long the simulation takes (less than 4 min for the current setup) and can guide students' expectations.
   » One can also individually run the cells containing the code and can also experiment with the values as each line of the code has a clear detailed comment explaining the intent of the same.
3. Once you are happy that the lab runs successfully, it may then be used with students. Detailed simulation information is contained within the Getting_started_with_PyMAPDL.ipynb file.
4. Depending on how much you want students to engage with the Python code, you may wish to hide the code cells (in Jupyter Lab, do this by selecting a cell and then clicking the blue vertical bar that appears to the left of it).

7. Expected Outcomes

Each cell explains the code with a detailed description as markdown and comments beside each code line. The following visualizations are printed below the respective cells on successful execution. Results such as mesh quality and tolerance, reaction forces are printed as text right below the cells on a successful solution.

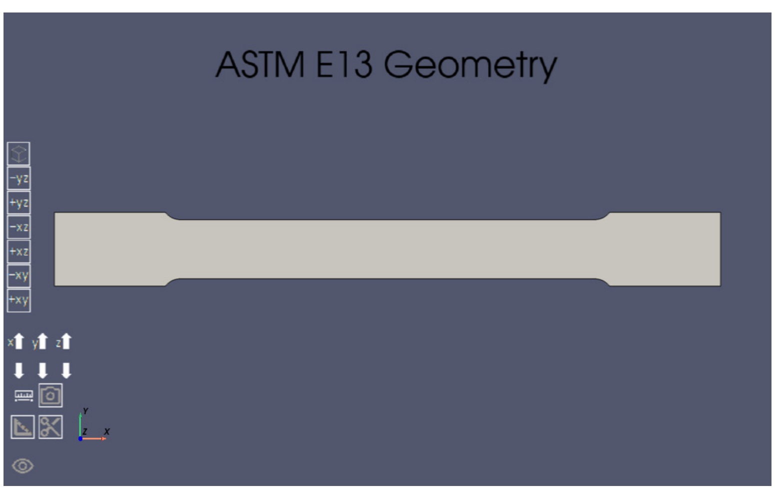

Figure 1: Generated Geometry for ASTM E13 standard

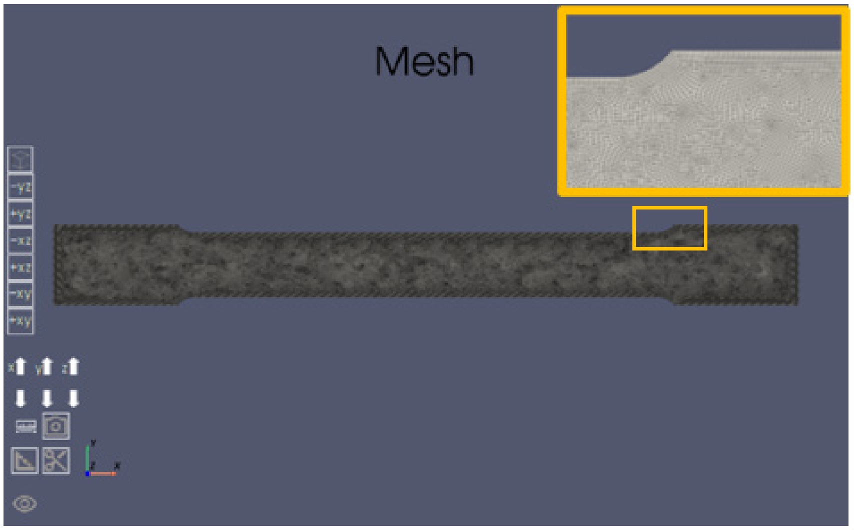

Figure 2: Generated quad dominant mesh for 1mm element size

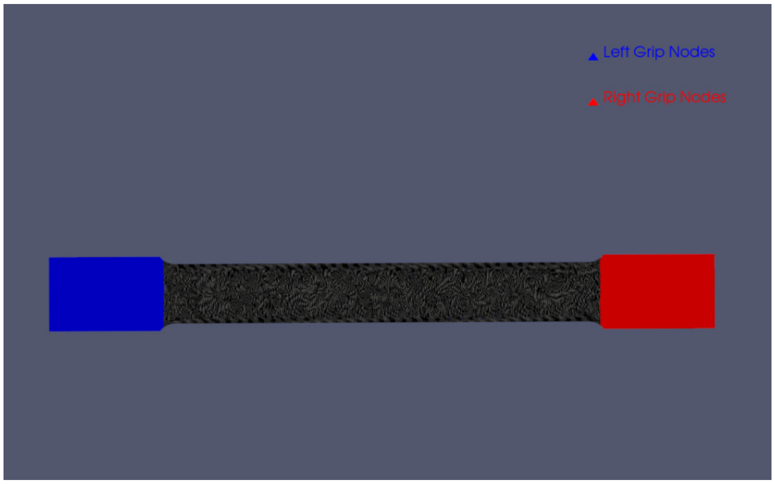

Figure 3: Applied boundary conditions replicating the tensile loading with blue portion as constrained while red as displaced nodes

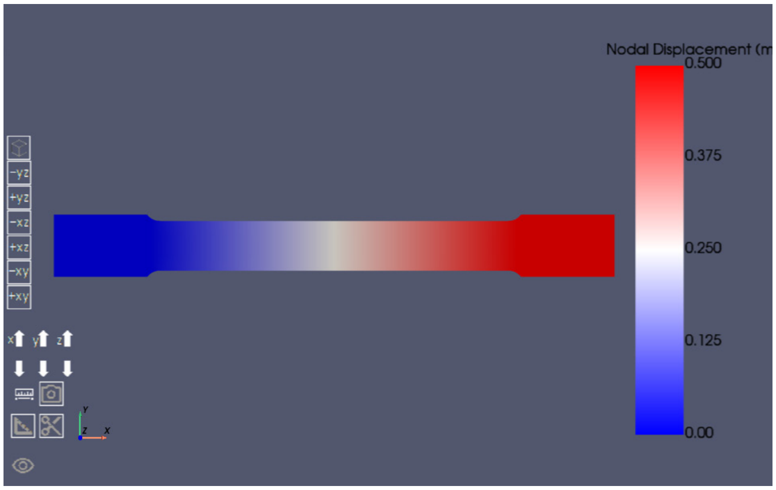

Figure 4: Displacement result without the element visualization

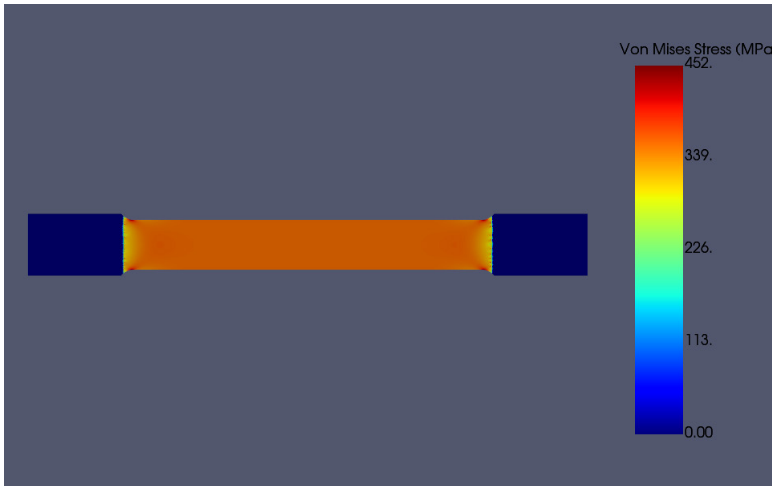

Figure 5: Equivalent stress result extracted in a user-defined way without the element visualization

If the user faces any error regarding the Jupyter notebook and the visualization interaction, please use the Getting_started_with_PyMAPDL.py, a python file as an alternative.

8. Optional Exercises

As a part of learning and experimentation,

1. Geometry parameters to visualize the changes and their effect on stress distribution,
2. Displacement and its effect on stress distribution and force reaction
3. Mesh size and shape
4. Method for extraction of results and try defining a new result entity

For learning more on result visualization uncomment the portions from the respective cells to visualize the results with the element/ mesh visualization.

9. Python Support

Documentation for Jupyter can be easily found online. Documentation for PyMAPDL– and other PyAnsys libraries (PyAnsys — PyAnsys) – can be found on the Ansys Developer Portal, along with examples, support articles and a user community.

10. Acknowledgments

This resource would not have been created without the assistance of Dr. János Plocher.

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