

Introduction to: Ansys GPT and Engineering Copilot

Ted Sperry Ph.D.
Application Engineer

8/7/2025



Agenda

- Ansys GPT
- Ansys Engineering Copilot
- Requirements for Access
- Questions

The logo for Ansys GPT, featuring the word "Ansys" in white with a yellow diagonal bar through the 'A', followed by "GPT" in white text inside a black rectangular box.The logo for Ansys Engineering Copilot, featuring the word "Ansys" in white with a yellow diagonal bar through the 'A', followed by a yellow slash and the words "ENGINEERING COPILOT" in white capital letters.

Ansys GPT Virtual Assistant



A **virtual assistant** designed to assist users with Ansys specific knowledge.



Provides **high quality responses in minutes** by tapping into a vast database of Ansys knowledge.



Holds conversations in **multiple languages**.



Scalable & secure deployment to customers on Ansys Innovation Space.



Built using Azure OpenAI Service.



Data Security and Governance



Uses Ansys owned or licensed data, which is internally vetted for quality prior to upload.



Customer queries and responses are neither stored nor used to train the model.



Ansys only tracks quantity of tokens used to identify abuse or misuse of the system.



Leveraging Azure Open AI Services and associated enterprise scale, Responsible AI and Security.



Ansys GPT Differentiators



Generates responses true to the relevant granular Ansys knowledge.



Minimal hallucinations.



Reliable, efficient, user friendly.



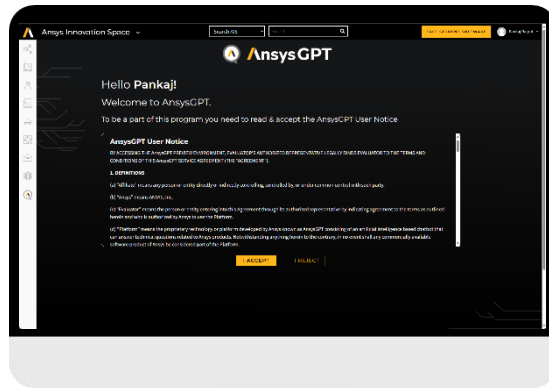
Prompts or responses are neither stored nor used to train the system.



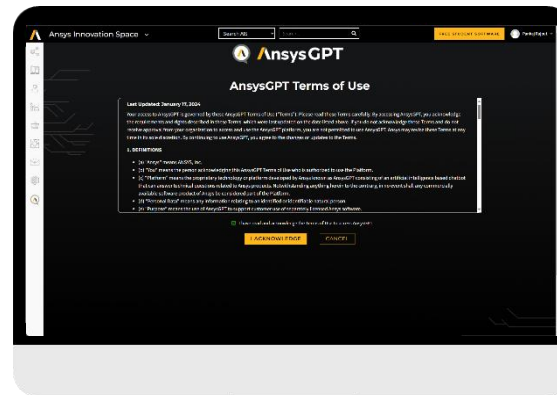
Getting Started with Ansys GPT



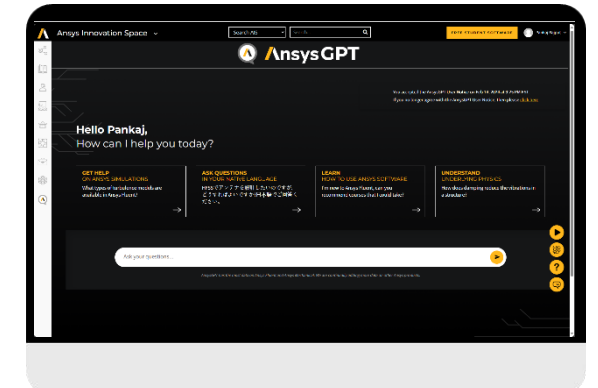
- Please ensure you have a valid login to Ansys SSO – **DRD Can assist with this setup**
- First time User Interface:
 - Login from **ansys.com/gpt** or **ansysgpt.ansys.com** using same login
 - Read and acknowledge User Notice
 - Read and accept Terms of Use
- One-time step, except for updates to user credentials or new software versions



User Notice



Terms of Use



Start Using



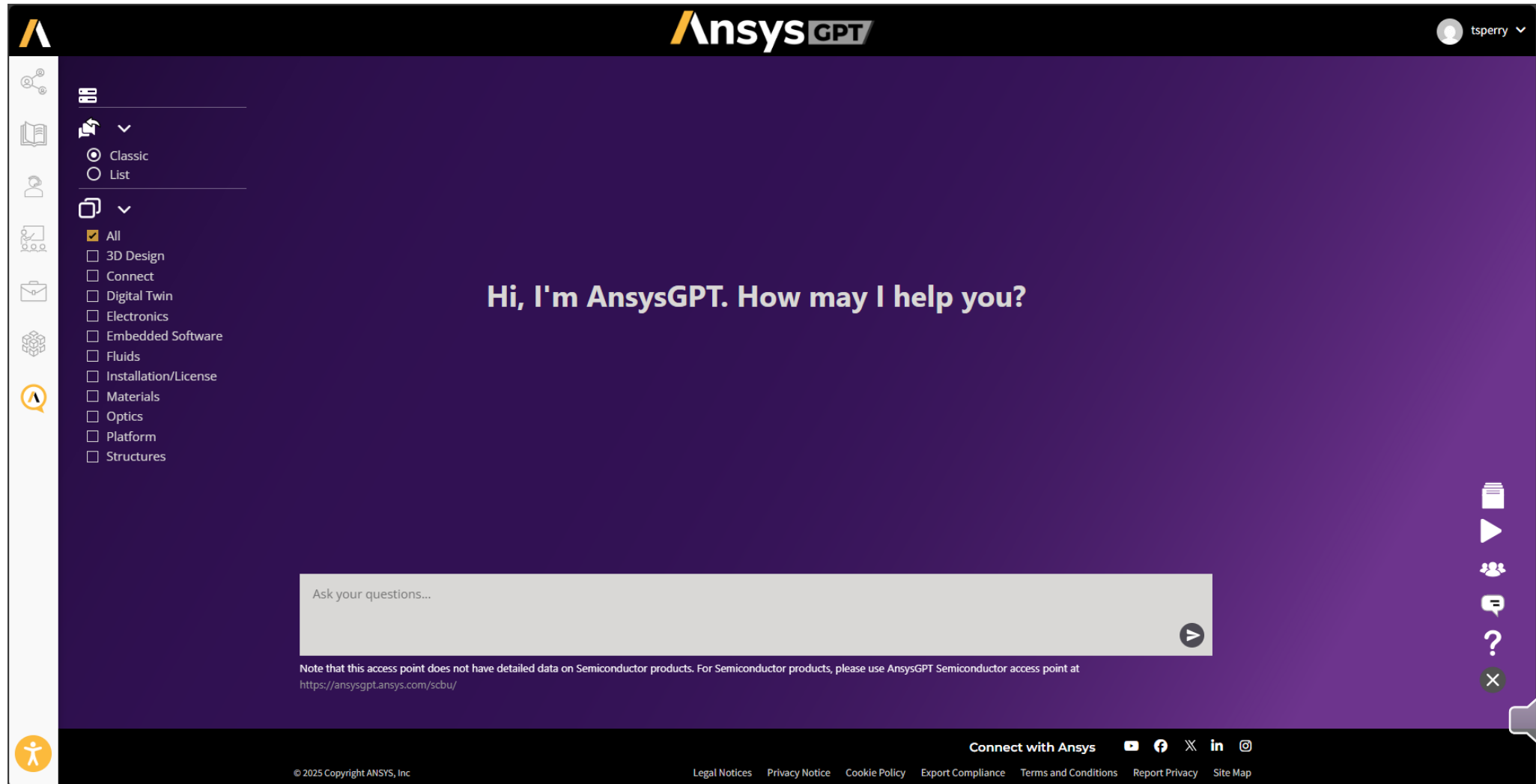
AnsysGPT Feedback and Help



- Ribbon on left side dictates the results that will be included in query
- Right side ribbon contains links to helpful resources for using AnsysGPT



Ansys GPT Demonstration





- ☒ Classic
- ☐ List



- ☒ All
- ☐ 3D Design
- ☐ Connect
- ☐ Digital Twin
- ☐ Electronics
- ☐ Embedded Software
- ☐ Fluids
- ☐ Installation/License
- ☐ Materials
- ☐ Optics
- ☐ Platform
- ☐ Structures

Hi, I'm AnsysGPT. How may I help you?

Ask your questions...



Note that this access point does not have detailed data on Semiconductor products. For Semiconductor products, please use AnsysGPT Semiconductor access point at <https://ansysgpt.ansys.com/scbu/>



+

🏠

📖

👤

📊

📁

🔍

🗨️

🏠

📖

👤

📊

📁

🔍

🗨️

1. 📖 3.01/4.0 ☒

2. 📖 2.95/4.0 ☒

3. 📖 2.89/4.0 ☒

4. 📖 2.84/4.0 ☒

5. 📖 2.75/4.0 ☒

6. 📖 2.70/4.0 ☒

7. 📖 2.67/4.0 ☒

8. 📖 2.67/4.0 ☒

9. 📖 2.66/4.0 ☒

10. 📖 2.66/4.0 ☒

What new features are available to the GPU solver in Fluent 2025R2?

In Fluent 2025R2, several new features have been introduced to the GPU solver. Here are some of the highlights:

- Boundary Conditions and Performance Improvements:** The GPU solver now supports boundary conditions such as Inlet Vent and Exhaust Fan. Performance improvements have been made for models like the S2S model with non-changing geometries/boundary conditions, Sliding mesh, and Flamelet Generated Manifold (FGM). Additionally, modeling electric potential is now supported, and you can track the current and peak memory of GPUs. The ability to enable asynchronous outputting for the native GPU solver is now a full feature, which can improve solver performance during calculations by up to a factor of 2. The Discrete Phase Model (DPM) features supported with the GPU solver have been upgraded from beta to release. 2
- Direct Postprocessing:** Direct postprocessing is available as a beta feature, allowing efficient transfer of field and mesh data from the GPU Solver directly to the Ansys Fluent renderer. This method is currently the default in the GPU Solver -lite mode and supports various postprocessing features such as Graphics objects, Mesh Display, Contours, Vectors, and more. 8
- Turbulence Models and Discretization Schemes:** The Large Eddy Simulation (LES) and Stress-Blended Eddy Simulation (SBES) turbulence models are supported, with options to modify parameters like Bounded Central Diff. Boundedness and Diffusion Coefficient. The Low Diffusion Central scheme for momentum is available, and various discretization schemes for flow equations are supported. 4

These enhancements aim to improve the efficiency and capabilities of the Fluent GPU solver, making it more versatile for complex simulations.



What are the benefits of using the Direct Postprocessing feature in Fluent 2025R2?

How does the support for new boundary conditions improve the GPU solver's performance?

Can you explain the significance of the Optimized LES Numerics option in Fluent 2025R2?

Ask your questions...

I



Note that this access point does not have detailed data on Semiconductor products. For Semiconductor products, please use AnsysGPT Semiconductor access point at <https://ansysgpt.ansys.com/scbu/>

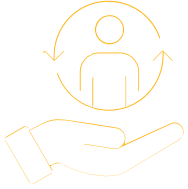
CLEAR CHAT

How to Use Ansys GPT



- Ask full phrased questions rather than keyword search.
- Ask follow-up questions in the same conversation.
- Use 'CLEAR CHAT' before asking a question on a new topic.
- If the answer provided doesn't seem to be relevant or appears inaccurate, ask the question in a different way or ask the bot to clarify which would allow AnsysGPT to refine its response.
- Specify the product in your questions if you are looking for product specific information, (e.g. Fluent 2024 R1)
- None of the conversation are stored or used for training AnsysGPT.
- Avoid using AnsysGPT for general-purpose tasks, like summarization, translation, email composition, script or code generation, etc.





50+ years of Technical Support knowhow in the product



Seamless access within one window



Side-by-side learning and support **powered by AI Agents**



First platform among **Ansys engineering** simulation peers



Ansys Copilot Components



Home

Search **ALL Ansys websites**



Learning

Access **800+** Ansys Innovation Courses



Knowledge

Thousands of **Knowledge Articles**



Forum

Global Forum of Ansys Users



Support

Create and Track Support Cases



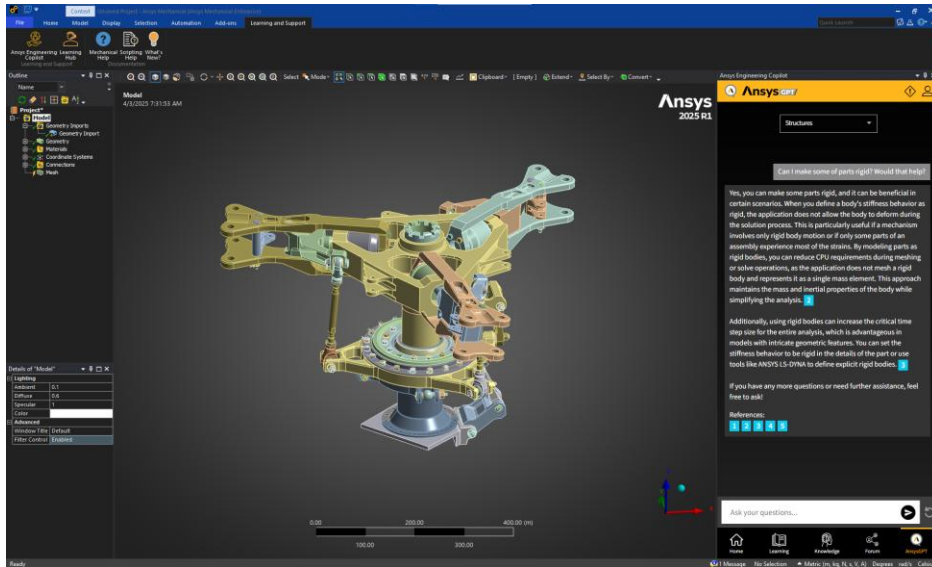
AnsysGPT

24/7 Multilingual **Virtual Assistant**

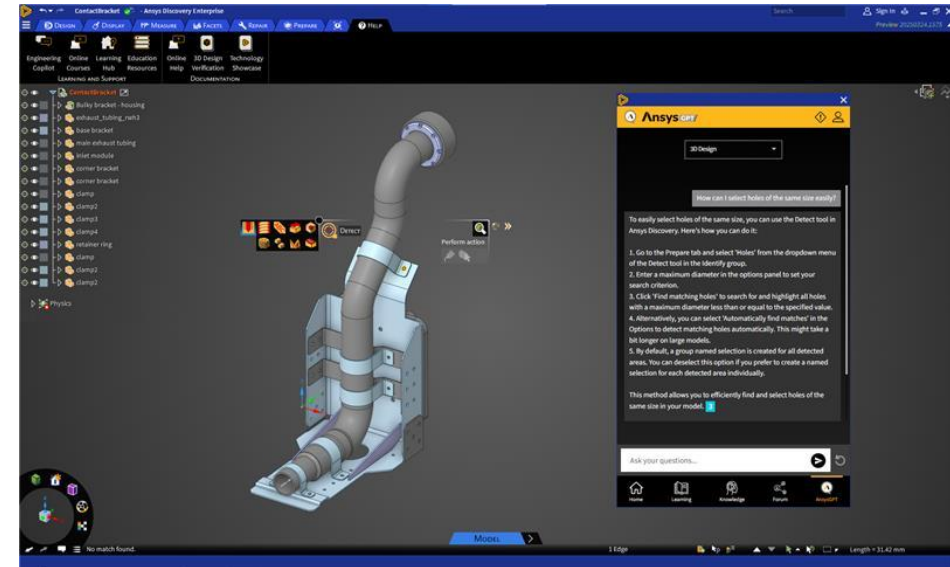


Ansys Engineering Copilot: Current Implementation

Mechanical



Discovery



Mechanical

Discovery

Fluent

Electronics
Desktop

SPEOS

Scade One

Optislang

Lumerical



Demonstration of Ansys Copilot

The screenshot displays the Ansys Engineering Copilot interface. The top menu bar includes File, Domain, Physics, User-Defined, Solution, Results, View, Parallel, Design, Parametric, and Learning and Support. Below the menu bar, there are tabs for Learning & Support and Documentation. The main workspace is divided into several panels:

- Outline View:** Contains a Case View section with a Filter Text input and a tree view showing Setup, Solution, Results, Parameters & Customization, and Simulation Reports.
- Task Page:** Contains a General section with Mesh (Scale..., Check, Report Quality, Display..., Units...) and Solver (Type: Pressure-Based, Density-Based; Velocity Formulation: Absolute, Relative; Time: Steady, Transient) options. There is also a Gravity checkbox.
- User Window 1:** The main workspace area, currently showing a 3D model of a mechanical part.
- Console:** Displays system messages, including "Graphics driver currently in use: dx11" and "Posting ANSYS Product Improvement Program startup data Done."

On the right side, there is a sidebar for the Ansys Engineering Copilot, featuring a Home button, a Recommended for you section with links to Fluids Courses, and a Search AIS section with a search bar and a magnifying glass icon.

Clipboard

Mode

Sketch

Sketch

Edit

Intersect

Create

Body

Assemble

Automate

Paste

Sketch

Select

Pull

Move

Fill

Combine

Split Body

Split

Project

Shell

Offset

Mirror

Equation

Cylinder

Sphere

Tangent

Align

Orient

Rigid

Gear

Anchor

History Tracking

Script Editor

Extension Builder

mixer-orig-complete*

body of influence

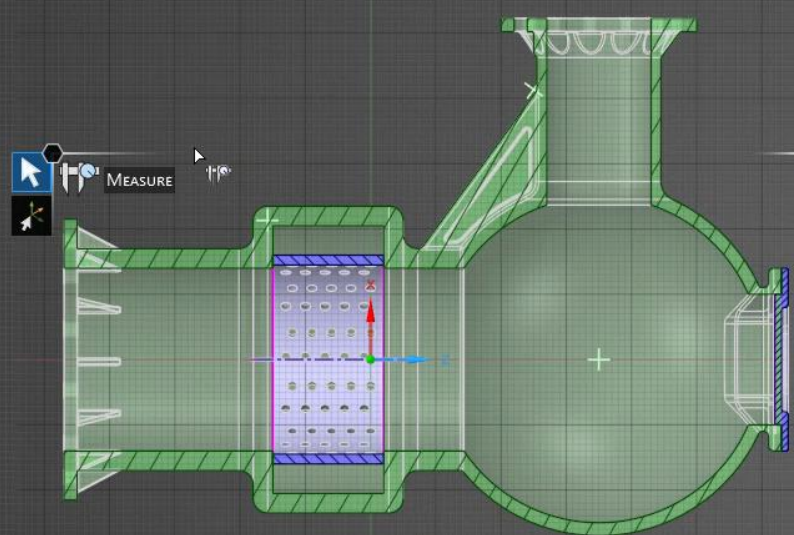
geom

lock

drum

mixer

Physics



MEASURE

Precision 4

Angular precision 2

Units Millimeters


Show XYZ vectors

MODEL

16

8/7/2025

Click on an object to measure. Ctrl+click to add more objects to the measurement.



Requirements for Ansys GPT and Engineering Copilot Access

- Both Ansys GPT and Engineering Copilot require users to have an Ansys account (SSO).
 - DRD can provide information required for this.
- Ansys Engineering Copilot requires users have Ansys 2025 R2 installed on their machines.
 - No software updates required on the license server.
- Ansys Engineering Copilot requires a new product added to the company account and a new license key created.
 - **DRD will assist with this process** of adding license to existing servers.
 - AnsysGPT AI+ License is **FREE**

AnsysGPT AI+

Act-200105-ER-40938-ANSYS-0TPf00000VGi8IMAD-SF



Questions?

Ansys GPT

Ansys / ENGINEERING COPILOT

- Start using these tools now – go try it out!
- Providing feedback will help improve Ansys GPT's responses
- Copilot will be added to more applications in the future
- DRD is Here to help users access and make the most out of these tools

Thank you for your attention!

If we run out of Q&A Time, feel free to contact **Support@drd.com**

