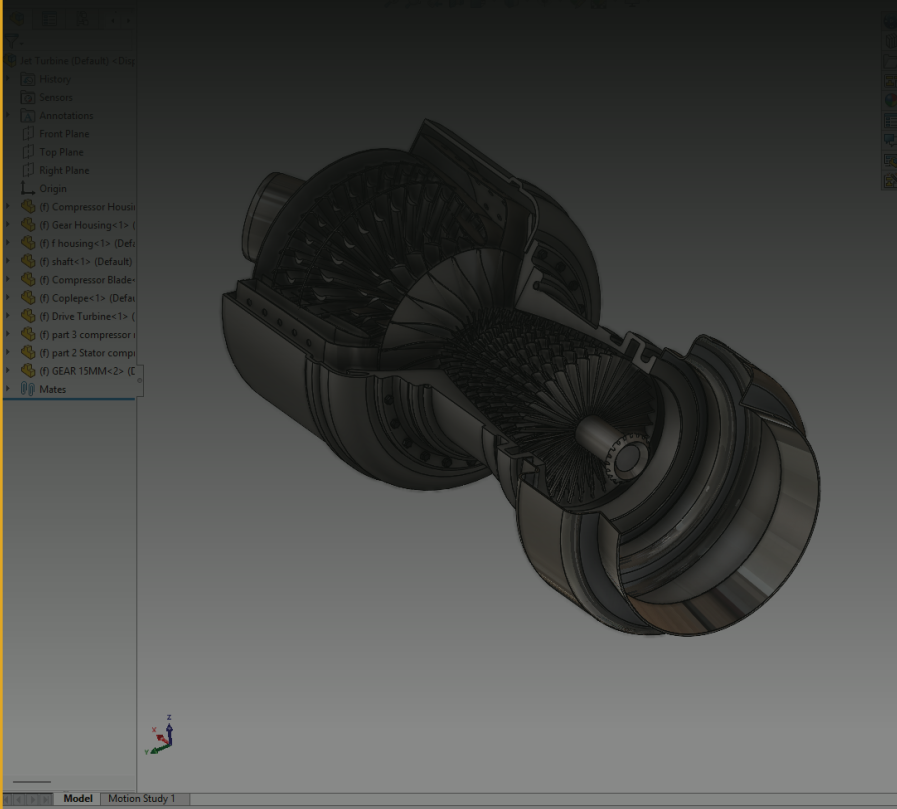


User guide


AI CAD Copilot
(v1.2.0)



Assistant

Hi! I'm MecAgent. You can ask me to create any automation you need, for example batch exports, auto-fillets, or mass renaming of assemblies. Let me free up your time and save you hours for real engineering.

Apply Fillet on X Axis Bulk Export DXF Multi-Body Part Coloring

Create an automation that... 

 Automations  Experimental

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Introduction

Introduction

What MecAgent is?

MecAgent is an **AI CAD Copilot** that generates CAD software **automations** from **natural language prompts**. You describe the automation you want, and MecAgent builds the script, provides dialogs when needed, and executes it directly inside the CAD software.

What MecAgent is not?

MecAgent **does not generate complex 3D geometry from text**, and it does not replace designers. It focuses on automating repetitive, low-value tasks using scripts. Any advanced 3D geometry generation demos you might have seen showcased online are experimental and still in development.

Introduction

What is possible?

- Processing large batches of CAD operations
- Modifying drawings at scale (title blocks, annotations, sheet names, properties, view settings)
- Managing assemblies (component names, repetitions, suppression state, basic positioning logic, custom properties)
- Exporting, renaming, cleaning and reorganizing files
- Measuring geometry and extracting dimensional or metadata information
- Creating dialogs and guided workflows
- Creating the first step of a drawing
- Create simple CAD parts in STEP and STL using natural language
- Get mechanical engineering answers for specific questions
- Perform simple tasks in the software using natural language (create parametric parts with a feature tree, change materials, translate names of parts, assemblies, and features)
- Find standard parts using natural language

What is not possible?

- Generating complex 3D geometry from text
- Creating advanced drawings with GD&T and specific views
- Human level reasoning across ambiguous instructions
- Create complex drawings with GD&T
- Fine-tune drawing creation using natural language

Why?

The intelligence of MecAgent happens during the creation phase. It takes a simple natural language request and transforms it into a fully structured automation, often involving thousands of lines of logic, without you needing to write or review any code.

Once generated, the automation runs as a standard script with no AI involved. The thinking happens at creation time. The execution is fast, predictable and code-driven. If you are still unsure what scripts can do and where their limits are, see Section 7 – Common Automation Examples for practical use cases.

Introduction

Why does MecAgent outperform standard AI chatbots ?

General AI tools like ChatGPT, Gemini, Claude or Copilot are not made for CAD scripting generation. They often guess, skip steps, or create code that doesn't actually work in CAD software. MecAgent is different. It was trained specifically on real CAD workflows and hundreds of thousands of CAD software automations, so it understands how engineers actually work.

Instead of improvising, MecAgent follows a controlled process to build a clean and reliable script every time. This is why the automations it creates run correctly in the CAD software, while general AI tools usually fail on anything technical or CAD related

Introduction

Requirements



Compatible with the latest SolidWorks versions (2026), all the way back to SolidWorks 2017.



Compatible with the latest Inventor versions (2026), all the way back to Inventor 2020.



Internet connection required.

Introduction

How to Download



1. Go to our Website : <https://download.mecagent.com/>
2. Log in, or create an account if you don't already have one

Full Name

Email

Password

Confirm Password

I agree to the [Terms of Service](#)

Sign up

Already have an account? [Log in](#)

support@mecagent.com

3. Click on the «Download for Windows» Button :

Download for Windows

4. The file can take a few minutes to be downloaded

mecagent-copilot-v0.0.23.zip
↓ 218/861 MB • 1 minute left

5. Extract All the download ZIP files

Extract All...

6. Run the «mecagent-app-0.0.X Setup»

mecagent-app-0.0.23 Setup

7. Sign in with your MecAgent account.

8. MecAgent connects to SolidWorks or Inventor automatically. If the connection fails, click the Reconnect button

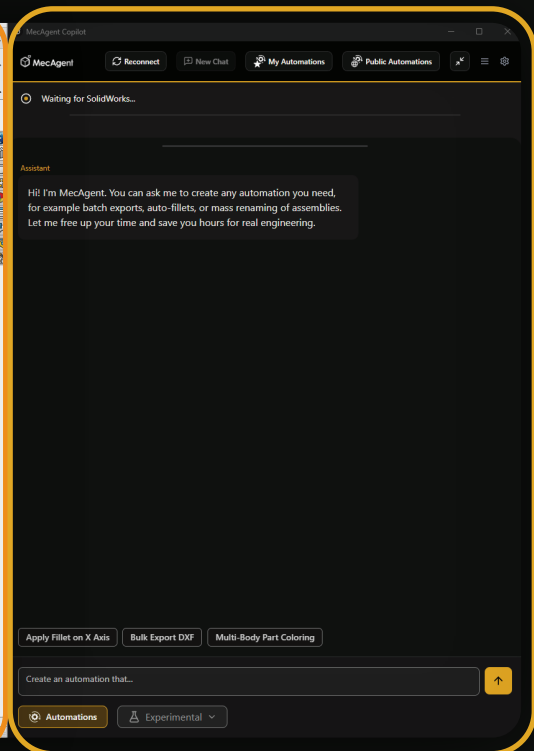
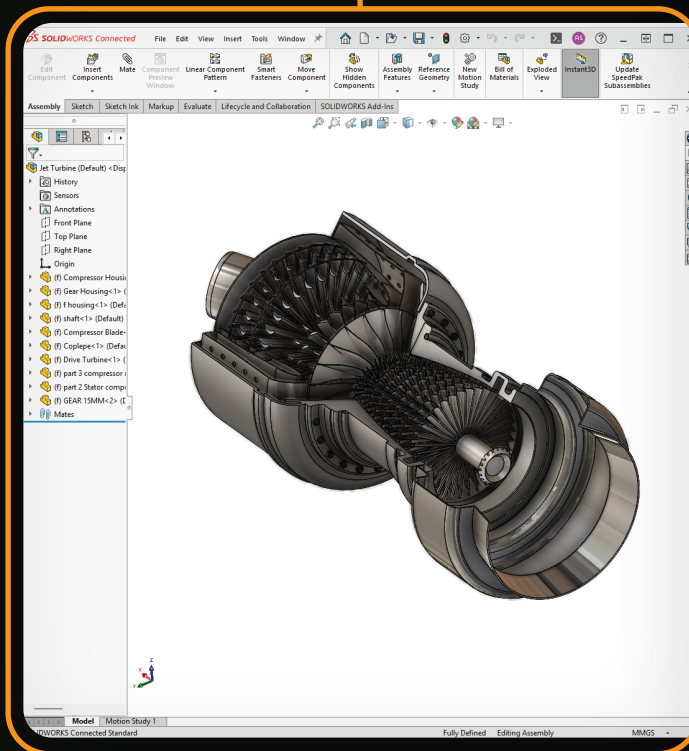
9. Finished, MecAgent Copilot is set up !! You can directly type in a chat to act inside the CAD Software

General Interface

General Interface

Standalone Software

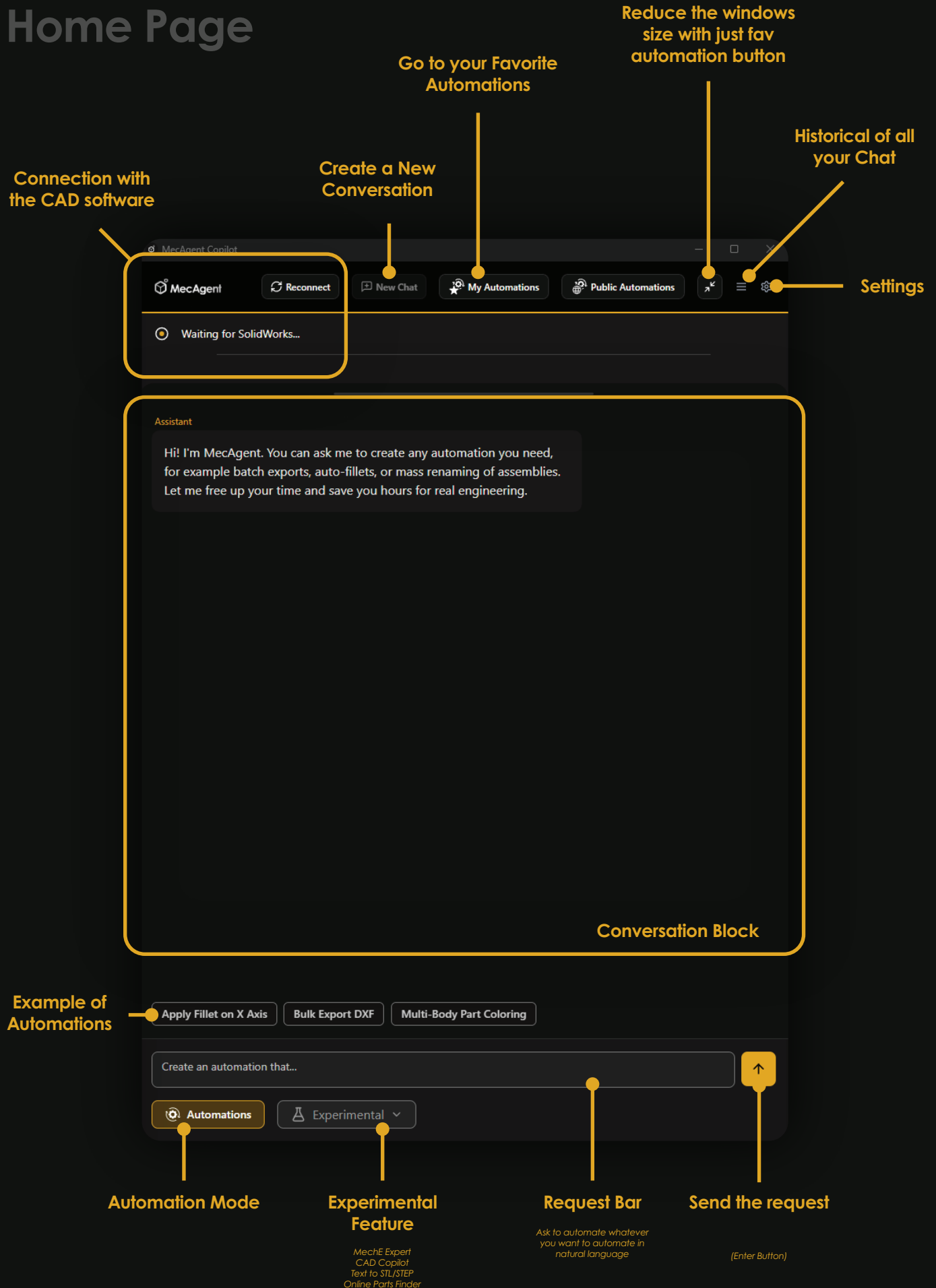
CAD Software
(Solidworks or Inventor)



Standalone -
MecAgent CAD
Copilot

Interface

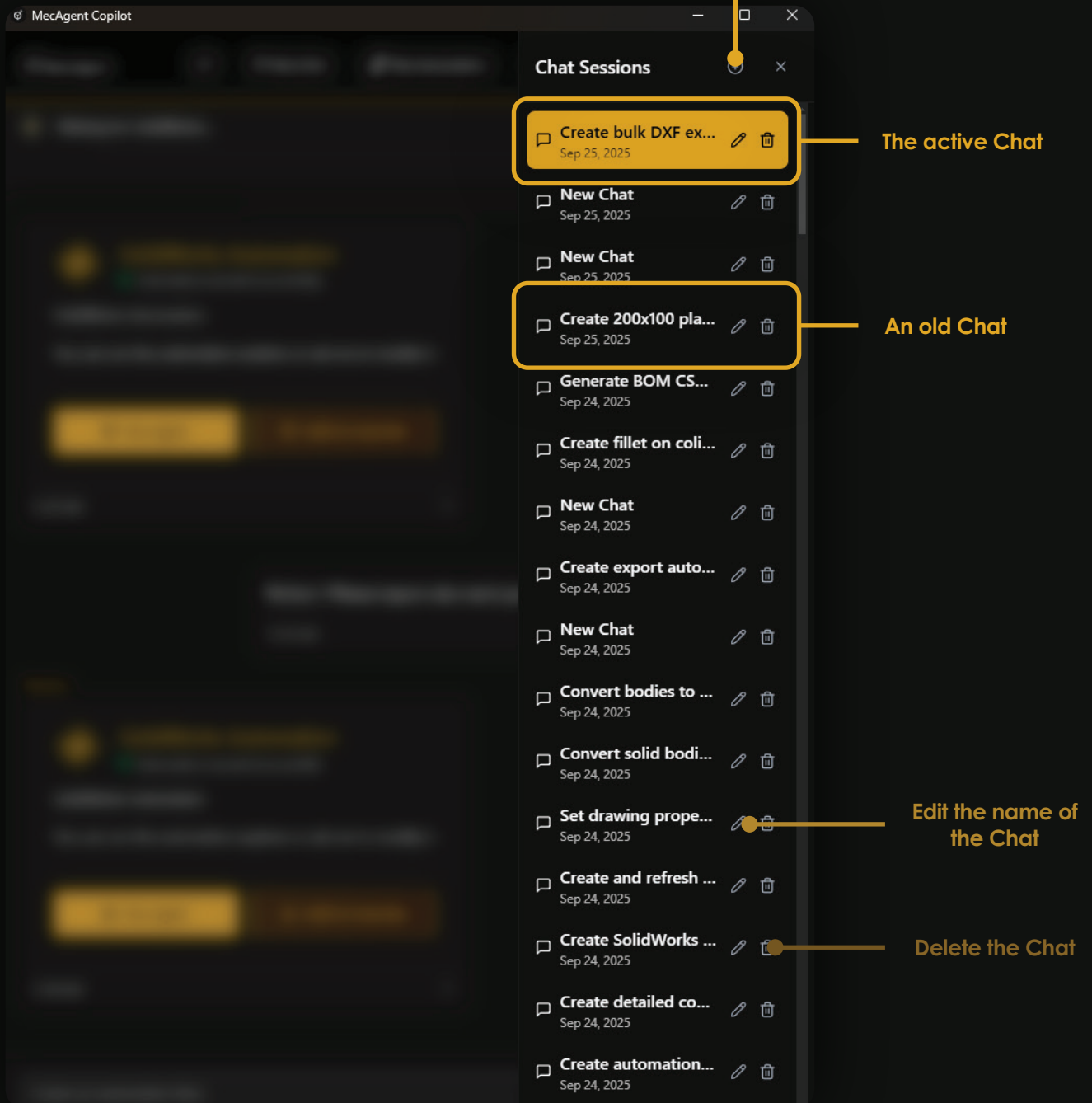
Home Page



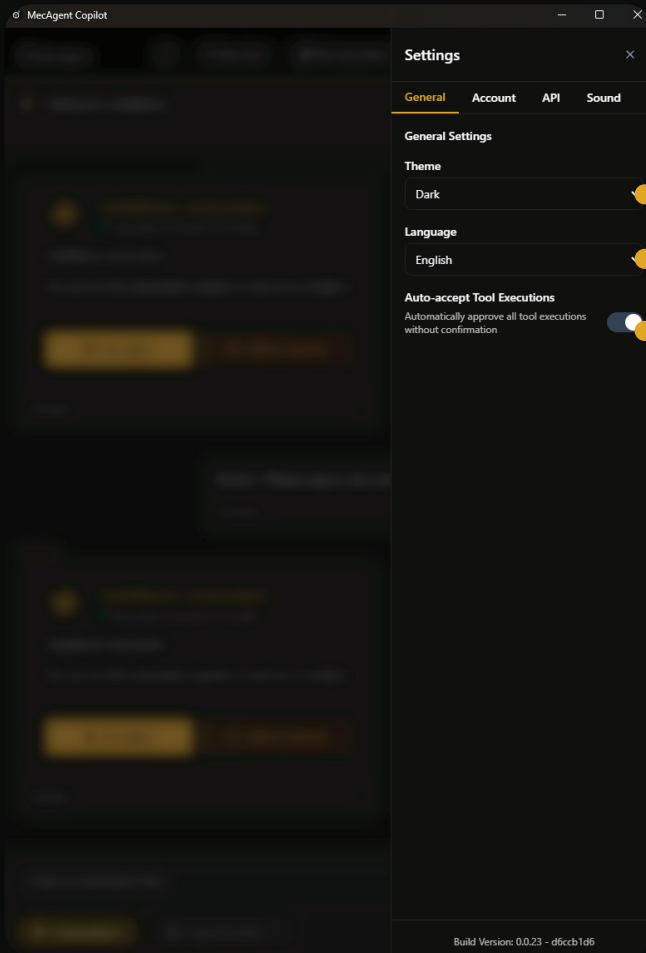
Interface

Chat history

Create a New Chat



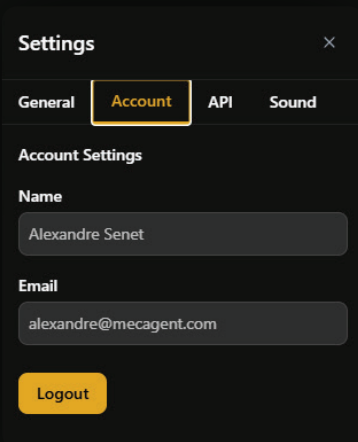
Interface Settings



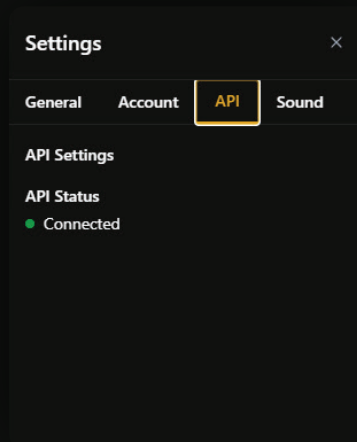
Change the Theme - Dark or Light

Change the Language

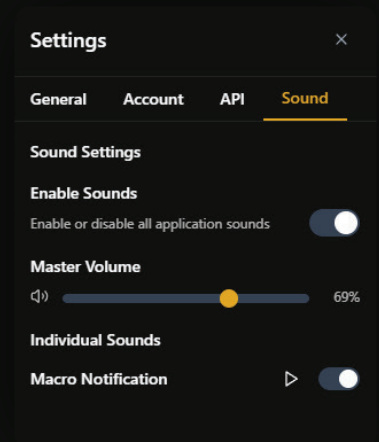
Auto-accept Tool Executions for CAD Copilot



Log out



See if you are correctly connected



Set notification sound for generated automations

Features & Capabilities

Features & Capabilities

General Features & Goals

MecAgent's goal is to add the latest AI technologies in all CAD software. Today MecAgent Copilot offers a range of features that may vary depending on the CAD software used and the development stage of the functionality.

Key Features :

- **CAD Macros AI Generation**
- **AI Mechanical Engineering Expert**
- **Drawing AI Generation**

R&D Features :

- **AI CAD Copilot**
- **Text-to-STEP/STL**
- **Standard Part AI Finder**

Features & Capabilities

Key Features

CAD Macros AI Generation

Create CAD Macros automations using natural language. Describe what you want to automate, and MecAgent generates and executes the corresponding script directly in your CAD software.

Automations can handle repetitive tasks across multiple parts, assemblies, or drawings, making them ideal for large-scale operations such as exports, renaming, and data processing. They also enable automated file management (creating folders, organizing files, structuring projects) and allow you to extract properties such as mass, dimensions, or metadata, with the ability to export results to Excel or other formats.

AI Mechanical Engineering Expert

When your request involves mechanical engineering knowledge, MecAgent can retrieve verified information directly from its engineering knowledge database.

This includes design guidance, engineering standards, calculations, and best practices used in real-world mechanical engineering workflows.

CAD Drawing AI Generator (Only on Solidworks)

Generate manufacturing drawings with AI directly from your CAD models. Automatically create views, dimensions, detail views, and standard drawing elements (*excluding GD&T and complex company-specific drawings, for the moment*).

Features & Capabilities

R&D Features

AI CAD Copilot (Only on Solidworks)

MecAgent acts as an AI CAD copilot to perform actions directly in your CAD software (simple text-to-CAD, renaming parts and features, updating properties, applying materials and appearances, translations, and accessing specific metadata).

Text to STL/STEP

Generate 3D geometry from text prompts. This feature explores early-stage geometry creation based on natural language descriptions. With this feature you can export STEP & STL files only (without feature tree).

Online Parts AI Finder

Search and retrieve existing CAD parts from external sources directly within your CAD environment.

This feature is currently under development and may not always return accurate or complete results. MecAgent is actively working with providers to expand access to standardized 3D part libraries.

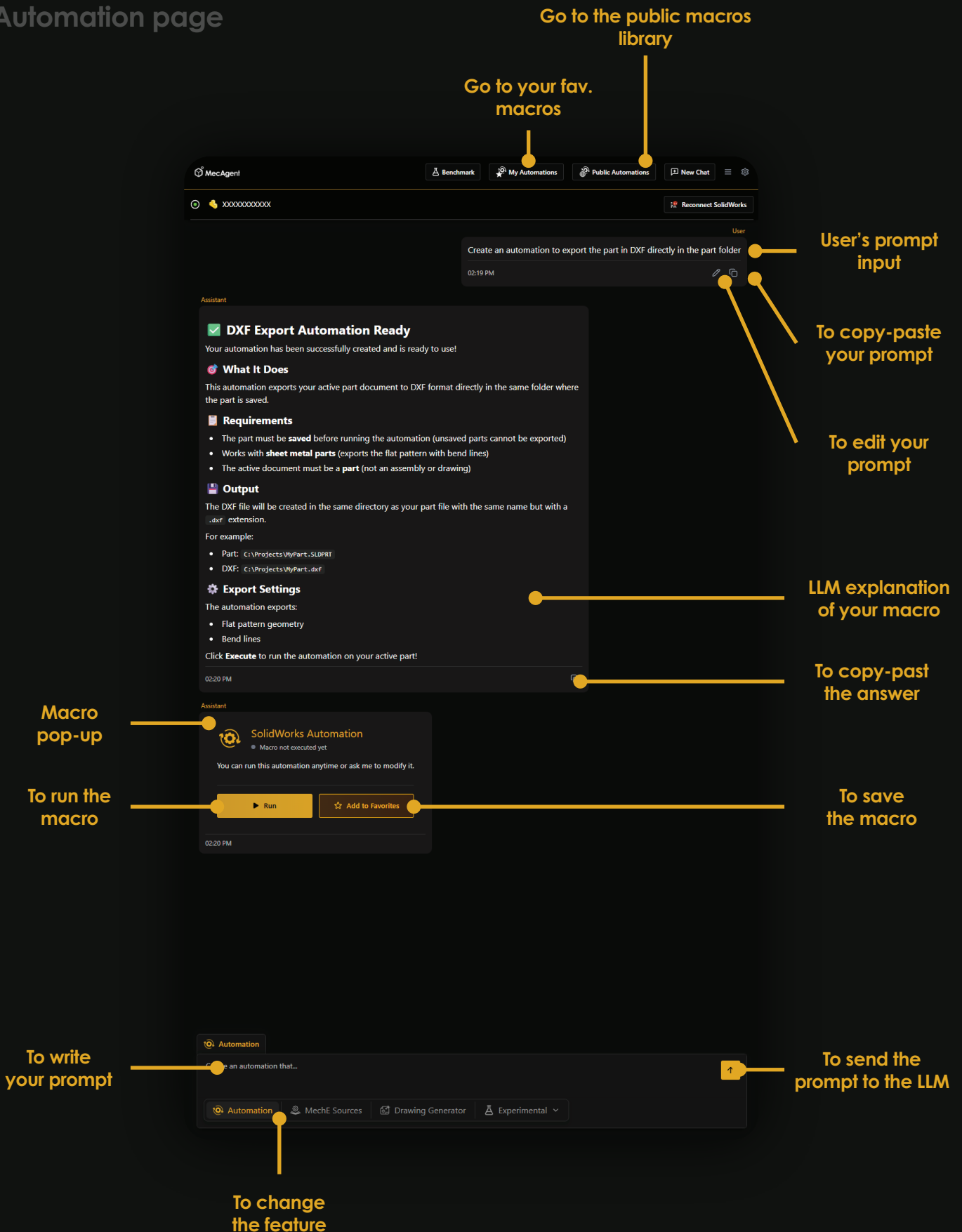
Key Feature

CAD Macros AI Generation

MecAgent allows you to create CAD macros with AI to automate anything in your CAD software (functions, bulk exports, specific drawings, appearance, sketching, saving, constraints, checking standards, etc.).

CAD Macros AI Generation Interface

Automation page



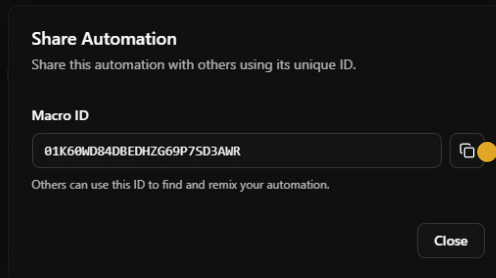
CAD Macros AI Generation Interface

My Automations

Reduce window with just fav macro button

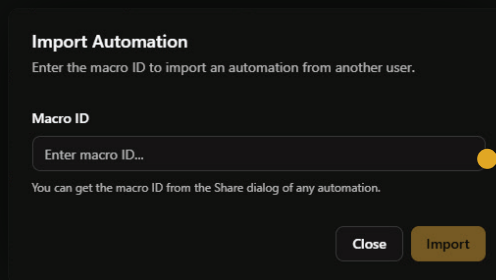
Share your macro

Run your macro



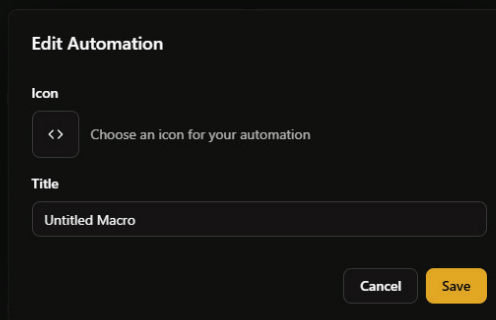
Copy the ID & Send the ID to your coworker

Receive the macro



Paste the coworker macro ID

Name & Icon Editing



CAD Macros AI Generation Interface

My Automations

Reduce the window size with just fav. macros buttons

Import a coworker's macro

Search for a specific macro

Go to the public macros library

The screenshot shows the 'My Automations' interface in a browser window titled 'MecAgent Copilot'. The interface includes a search bar, a 'Newest First' dropdown, and a grid of macro cards. Each card displays a macro name, a timestamp, the number of uses, and 'Run' and 'Remix' buttons. A yellow box highlights the 'Random Component Color Application f...' macro. Callouts with arrows point to various features: 'Search for a specific macro' points to the search bar; 'Import a coworker's macro' points to the 'Import Automation' button; 'Go to the public macros library' points to the 'Public Automations' button; 'Reduce the window size with just fav. macros buttons' points to the window control buttons; '1 macro' points to the highlighted macro card; 'Rename, Share & Delete your macro' points to the three-dot menu on the 'Automation for Sketch Repetitions and...' macro; 'Run your macro' points to the 'Run' button on the 'Bulk DXF Export of Sheet Metal Parts -...' macro; and 'Improve or modify your macros' points to the 'Remix' button on the 'Enhanced Part Turn Around Automation' macro.

CAD Macros AI Generation Interface

Public Automations

The screenshot shows the 'Public Automations' interface in the MecAgent Copilot application. The interface includes a search bar, a filter dropdown set to 'Recently Added', and a grid of automation cards. Annotations highlight key features: 'Search for a specific macro' points to the search bar; 'Filter the public macro library' points to the 'Recently Added' dropdown; '1 Public macro' points to the first card, 'Mesh Body Conversion and STL Export', which is highlighted with a yellow border; and 'Add the macro to your favorite macro' points to the 'Copy to My Automations' button on the 'Inserting Multiple Part Instances in an Assembly' card.

Search for a specific macro

Filter the public macro library

1 Public macro

Add the macro to your favorite macro

MecAgent Copilot

Public Automations

Browse and copy community-shared automations

Search public automations...

Recently Added

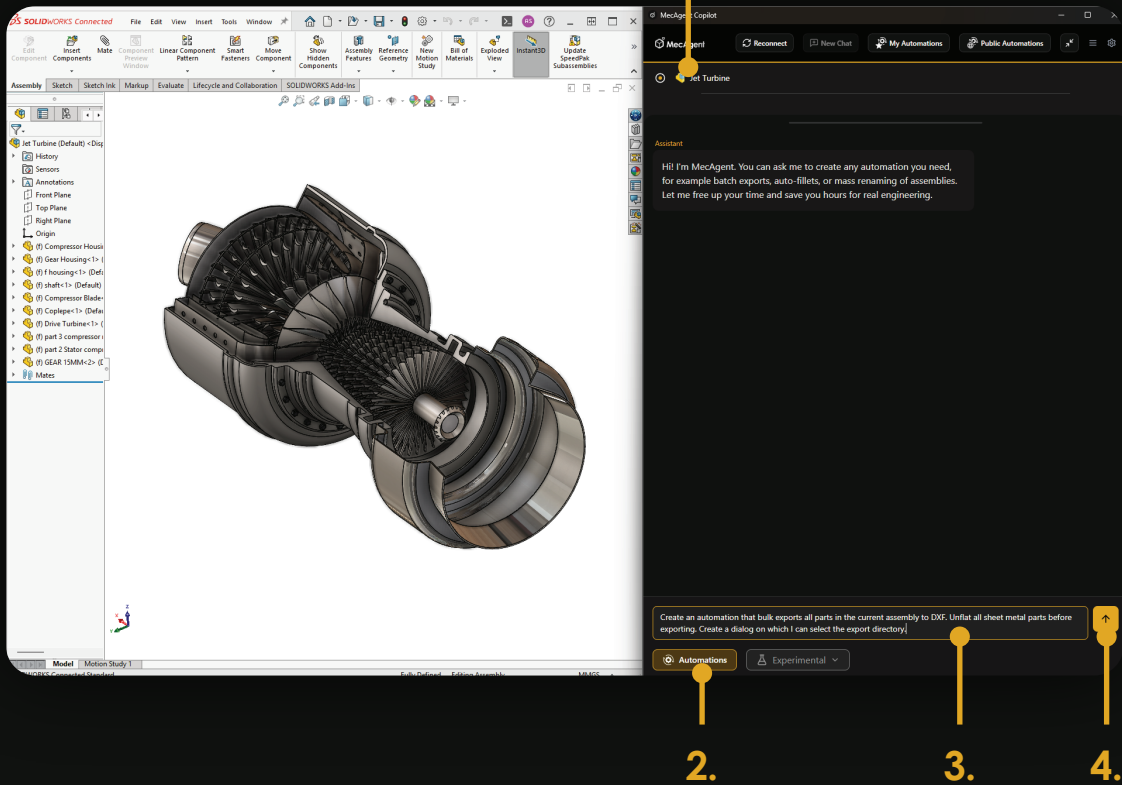
Showing 14 of 14 public automations

- Mesh Body Conversion and STL Export**
This automation code facilitates the conversion of solid bodies in a CAD part document into fin...
More
13 uses
Shared 1 days ago
Copy to My Automations
- Solid Body Slice Generator V2**
This automation code facilitates the creation of laser-cut friendly cross-sections from a selecte...
More
10 uses
Shared 1 days ago
Copy to My Automations
- Inserting Multiple Part Instances in an Assembly**
This automation facilitates the insertion of multiple instances of a specified part into an...
More
2 uses
Shared 2 days ago
Copy to My Automations
- Automation for Creating Construction Lines in a CAD...**
Creates two 200 mm construction lines inside a sketch (horizontal and vertical) centered on the...
More
8 uses
Shared 2 days ago
Copy to My Automations
- Solid Body Slice Generator**
This automation code facilitates the creation of laser-cut friendly cross-section slices from a...
More
38 uses
Shared 8 days ago
- Bulk DXF Export of Sheet Metal Parts - Assembly**
This automation tool facilitates the bulk export of sheet metal parts from an assembly into DXF...
More
6 uses
Shared 14 days ago

CAD Macros AI Generation

How it works ?

Create an automation



1. Verify that you are correctly connected to the CAD software.

(If not, go to section «Connection Issues with the CAD Software»)

2. Verify that you are in "Automations" Mode.

3. Write whatever you want to automate in natural language

(Prompt Examples :

To Bulk Export Sheet Metal : "Create an automation that for each sheet metal part, placing a flat view on the left and an unflattened view on the top right view on the top right. Export each drawing as PDF, DXF, and DRW, in a specific folder."

Mass Import Assembly : "Create a macro that inserts multiple instances of the same part into my assembly."

Auto Exploded View : "Write a macro that generates a default exploded view for the current assembly."

Isometric View Screenshot : "Create an automation that sets the view to isometric and saves a high-resolution screenshot."

Mass Properties Export : "Generate a macro that calculates the mass, volume, and surface area and saves the data in an Excel file."

BOM to Excel : "Generate a macro that exports the assembly bill of materials to an Excel file."

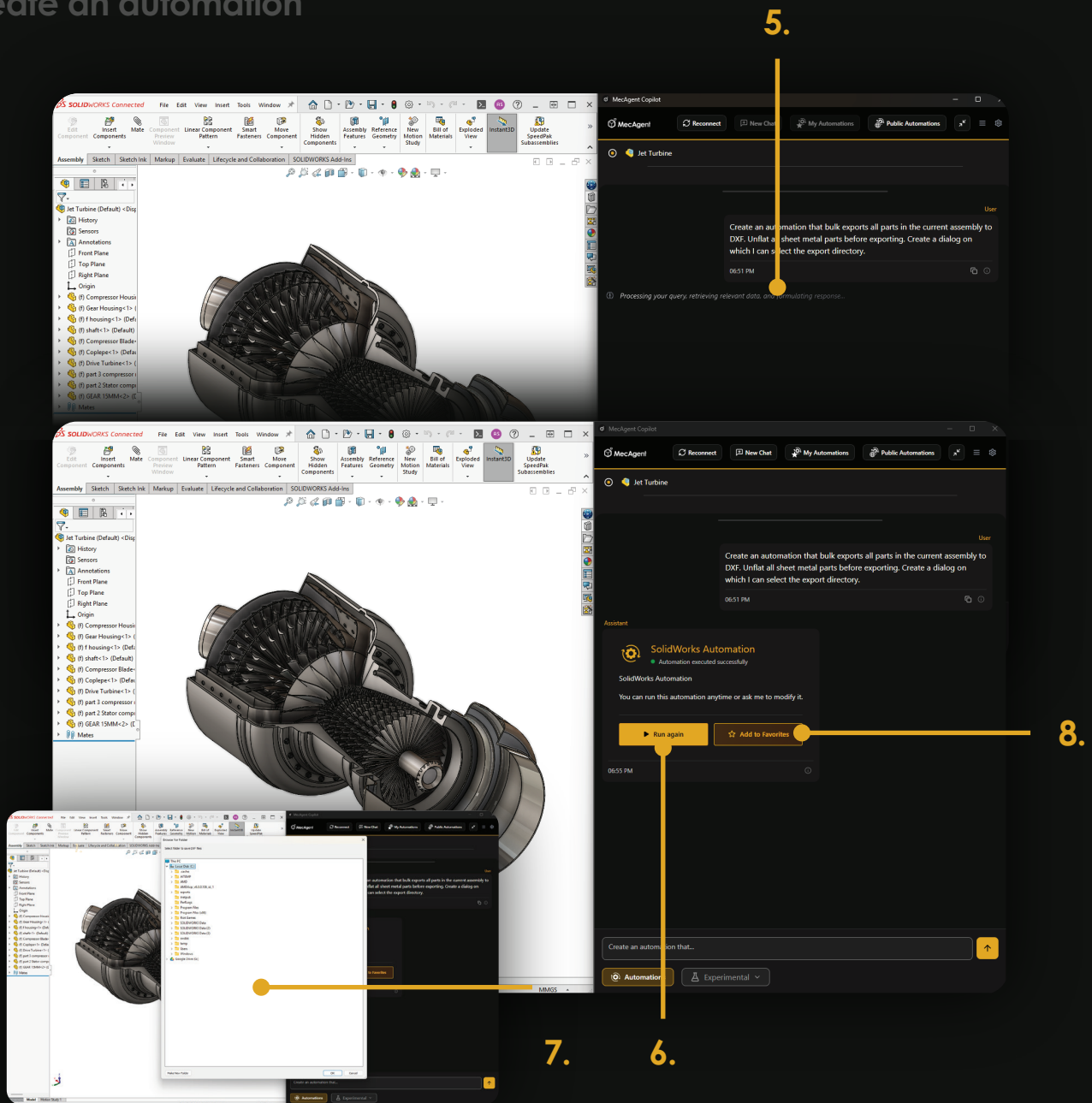
If you are still unsure what scripts can do and where their limits are, see Section 7 – Common Automation Examples for practical use cases.

1. Press Enter or click on the arrow button to send your request.

CAD Macros AI Generation

How it works ?

Create an automation



5. The AI processes your request to create the automation/macro. This takes between 30 seconds to 5 minutes. Once ready, you'll hear a sound notification.

6. Click "Run" to execute the automation instantly. After creation, it can be reused anytime, anywhere.

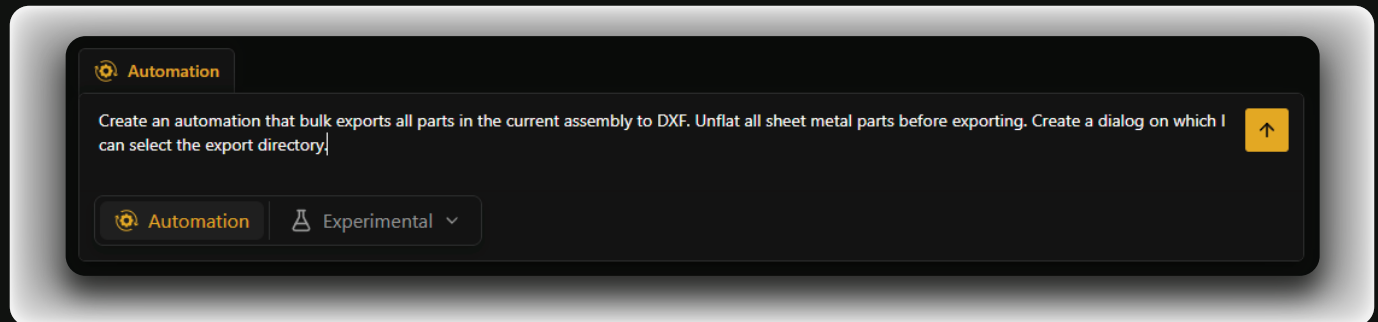
7. The automation runs.

8. If you are satisfied, click "Add to Favorites" to save it in your library.

CAD Macros AI Generation

How it works ?

How to prompt



Drawing creation:

- ✓ "Create an automation that goes through every sheet metal part in my current assembly. For each part, create a drawing, place a flattened view on the left and an unflattened isometric view on the top right."
- ✗ "Create all the drawings"

Mass Import Assembly:

- ✓ "Create a macro that inserts multiple instances of the same selected part into my assembly."
- ✗ "Insert new parts"

Isometric View Screenshot:

- ✓ "Create an automation that sets the view to isometric and saves a high-resolution screenshot."
- ✗ "Make a nice 3D render"

Mass Properties Export:

- ✓ "Generate a macro that calculates the mass, volume, and surface area and saves the data in an Excel file."
- ✗ "Extract the part's data in Excel"

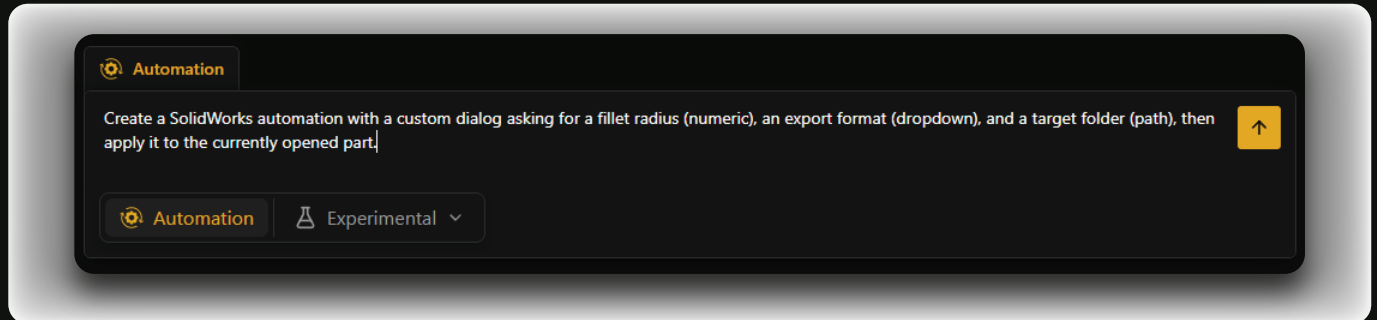
BOM to Excel:

- ✓ "Generate a macro that exports the assembly bill of materials to an Excel file."
- ✗ "Export to excel"

CAD Macros AI Generation

Advanced Prompting Techniques

Automations requiring user input



Automations can collect user input through a custom dialog interface. The best way to provide data to your script is to ask MecAgent to generate a dialog with the specific fields you need:

- **Numeric input boxes** (for example entering a fillet radius in mm)
- **Dropdown lists** (units, export formats, constraint types, etc.)
- **Text fields**
- **File or folder path inputs** (so the script can process all files in a directory for example)
- **Windows file chooser dialogs** (to select the part, assembly, or drawing to process)
- **Multi-field forms combining several inputs in a single UI**

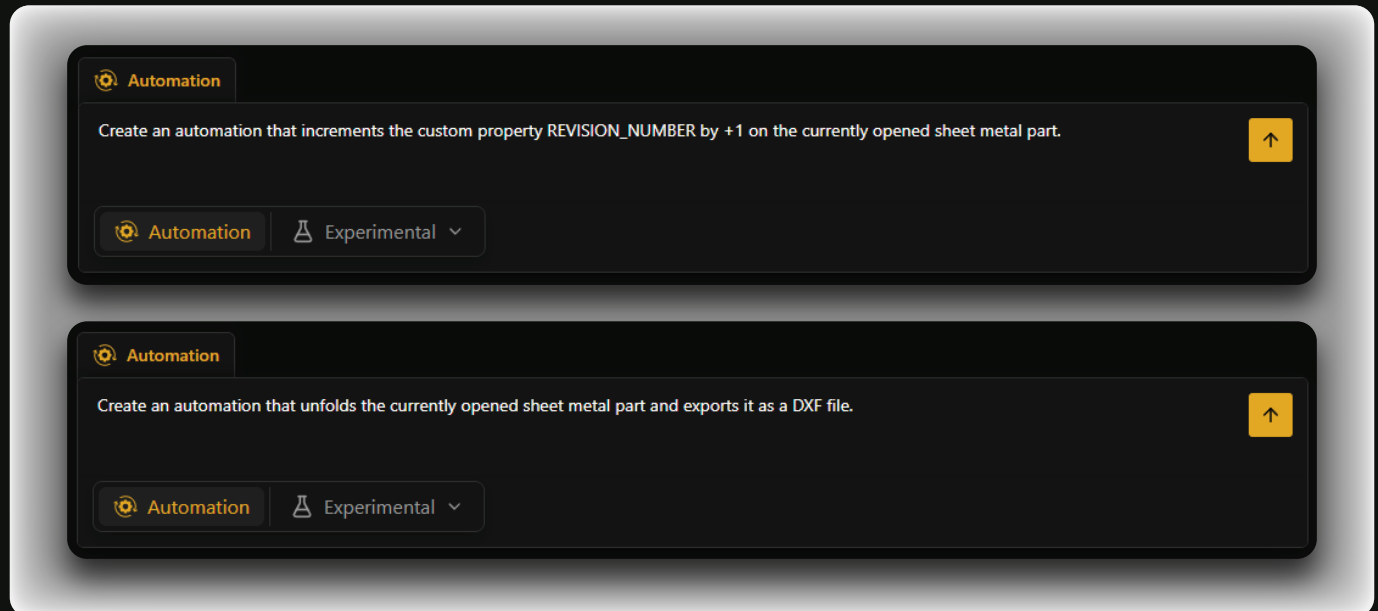
You can also refer to the active SolidWorks context by saying: "in my current assembly", "in my current part", "on the part I selected", "on the drawing currently opened", and MecAgent will automatically detect and use the right document.

This allows the script to receive all required information before running, making the automation far more flexible and powerful.

Advanced Prompting Techniques

Advanced Prompting Techniques

Breaking complex automations into smaller parts



For complex automations, start with the first step, get it working, then tell MecAgent to improve it by adding the next step, and so on. This makes the process easier to control and greatly increases the success rate. If a new step breaks the script and you want to restart from the last working version, simply save the good version (click on "Add to Favorites"), find it again in "My Automations", and click "Remix" to start fresh from a stable base.

For even more advanced workflows that involve multiple independent tasks, it's often better to **build** each main task **in a separate chat**, validate them individually, save them, and then use the **Merge feature** in **My Automations** to **combine them into a single final automation**.

Example:

If you want a macro that does the following for an opened sheet metal part:

1. Increment the custom property REVISION_NUMBER by +1
2. Unfold and export the part as DXF

You should split the work into two chats:

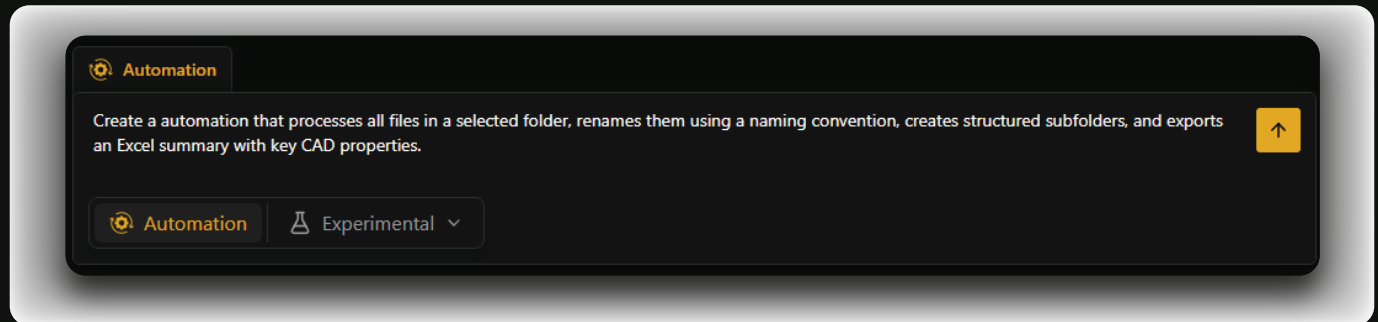
- Chat A: Create and validate the automation that modifies the custom property
- Chat B: Create and validate the unfold / DXF export automation

Once both are working, go to My Automations, click **Merge**, select the two scripts, and tell MecAgent how to **combine them into a single automation**.

CAD Macros AI Generation

Advanced Prompting Techniques

Combining SolidWorks and Windows operations



MecAgent scripts can **mix CAD actions** and **Windows file operations** in a single workflow. This lets you automate entire project pipelines from inside SolidWorks.

MecAgent can perform operations such as:
Rename files following a naming convention

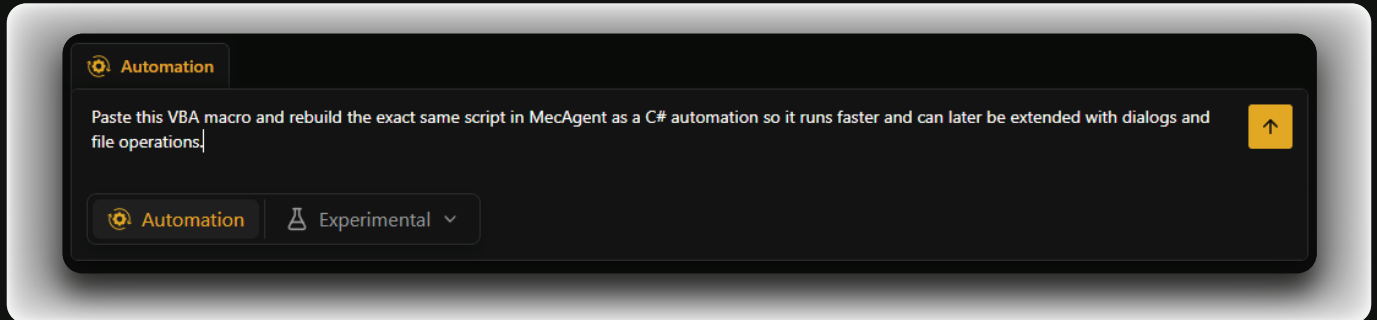
- **Create folders** (per part, per configuration, per assembly...)
- **Duplicate** or **move files** to structured directories
- **Clean** or **reorganize folders** automatically
- **Process all files** inside a selected folder
- **Generate Excel sheets** or **text files with extracted CAD data**

This is very powerful when automating manufacturing exports, QA pipelines, drawing management, project archiving, and batch processing.

CAD Macros AI Generation

Advanced Prompting Techniques

Rewriting an existing macro



You can **paste any VBA macro into the chat** and ask MecAgent:

“Rebuild this exact same script.”

MecAgent will convert it into its internal C# based automation environment.

This has several advantages:

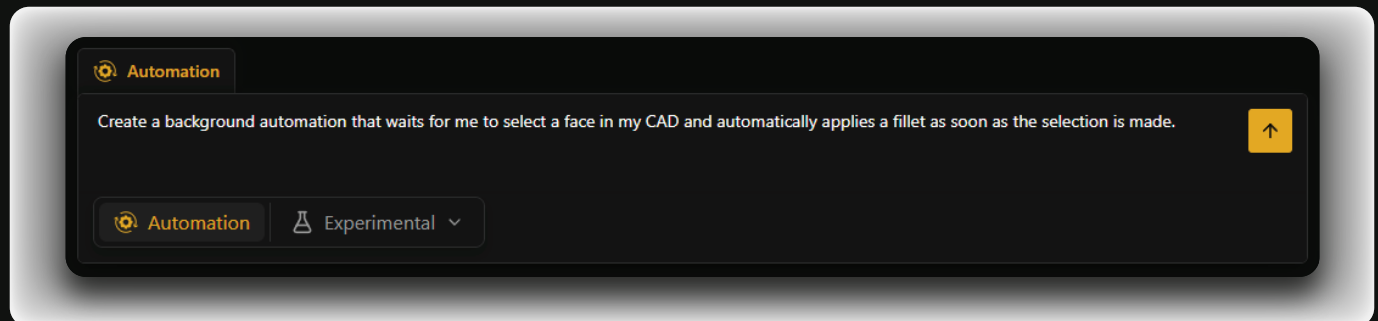
- **C#** executions are much **faster than VBA**
- They are **more stable** and **reliable**
- They support **more capabilities**
- You can **extend them** with dialog or file operations
- They run inside the MecAgent runtime, not the **outdated VBA environment**

This is the best way to modernize old legacy macros without rewriting them manually.

CAD Macros AI Generation

Advanced Prompting Techniques

Background listeners



Background listeners let MecAgent wait for a specific action inside SolidWorks before running the next steps of your automation. This is extremely useful when you want the script to react dynamically to what you do in the interface.

You can ask MecAgent to create listeners such as:

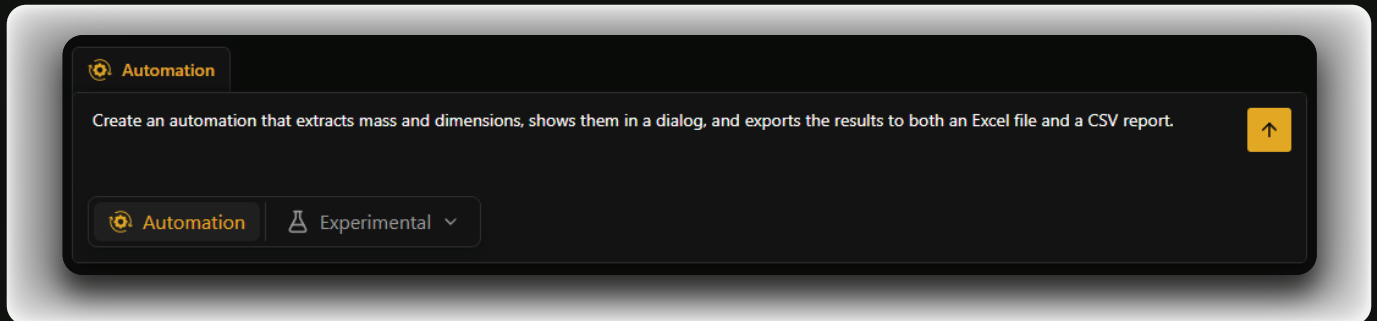
- **Wait for the user to select an entity** (face, edge, component, dimension...)
- **Wait for two Ctrl+clicks** and apply a rule or constraint right after
- **Wait for a specific selection type** (only bodies, only components, only sheet metal features)

This creates automations that run **"in the background"** and activate only when the required condition is met.

CAD Macros AI Generation

Advanced Prompting Techniques

Exporting or displaying results



Automations can output results in many formats depending on what you want to do. You can ask MecAgent to:

- Show the results in a **dialog box** (mass, dimensions, metadata, errors...)
- Generate an **Excel file** containing extracted information
- Output a **text file** (.txt, .csv, .json...)
- Write results into the **FeatureManager** or **custom properties**
- Save debug information into a **log file**
- Create a **summary report** after processing many files or components

This makes it easy to collect data, export measurements, document processes, or analyze assemblies at scale.

For debugging, you can also use the "Show execution logs" button

CAD Macros AI Generation

Debugging and Fixing

Fix an automation

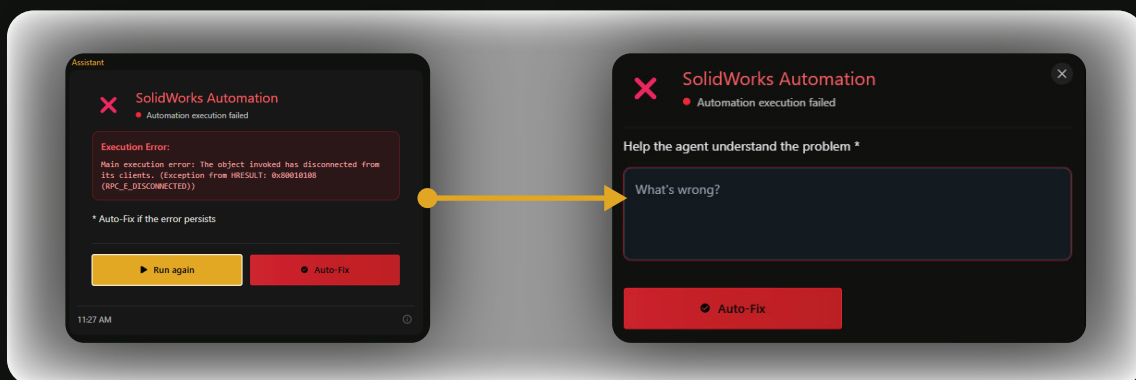
Sometimes the macro may fail due to an incorrect action, a missing step, a lost CAD connection, or an incomplete macro. Check the “Execution Error” dialog box to understand and fix the issue.

Using "Auto Fix"

Describe what went wrong and why (if you know) in natural language.

Asking for a debug dialog

Describing what went wrong and why in the auto fix chat input is even better when you're able to provide it, but it's not required.



Frequent user mistakes

- Asking for tasks that scripts cannot do (like generating complex 3D geometry or creating advanced GD&T drawings). See sections 1.1 and 1.2 for details.
- Use one chat conversation for one automation project only. Avoid mixing unrelated macros in the same chat, since this can confuse the AI, reduce performance, and significantly increase the risk of errors.
- Giving vague prompts
- Trying to put many automations in one conversation
- Forgetting to specify steps
- Not asking for dialogs
- Not saving intermediate working steps
- Avoid creating scripts that work with parts, assemblies, or drawings that are not saved, since SolidWorks scripts cannot fully interact with unsaved documents.

CAD Macros AI Generation

Common Examples

Point Cloud generation

Create 3D point cloud from XYZ coordinates

What a Bad Prompt Looks Like?

Create a point cloud from my XYZ coordinates list.

Why this prompt is bad?

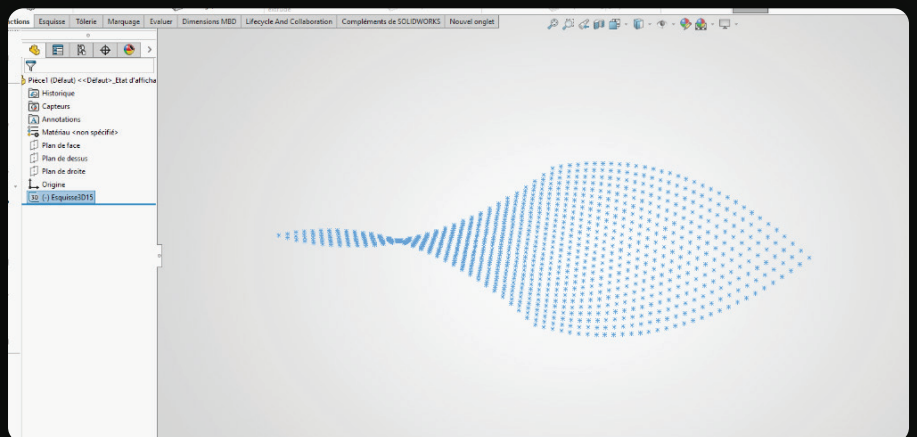
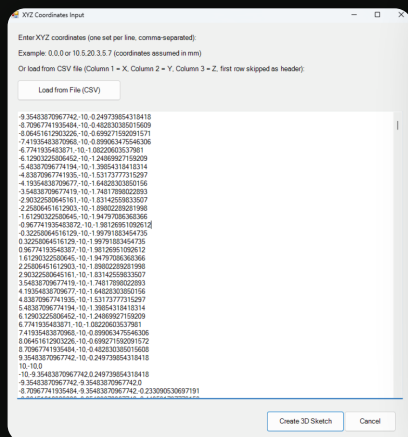
- It doesn't say which document is open.
- It gives no file format or way to provide the coordinates.
- It doesn't explain how the XYZ data is structured.
- It doesn't specify that a 3D sketch must be created to place the points.

What does a good prompt looks like?

In my current part, create a 3D sketch point cloud using XYZ coordinates imported from a CSV file. Show a dialog to select the CSV file and preview the coordinates before executing. The CSV file must have X in the first column, Y in the second, Z in the third, and the first row (header) will be ignored.

Why this prompt works?

- We tell Mecagent that a part is currently opened
- The input file format (CSV) is clearly specified.
- The prompt explains that the file will be selected through a dialog interface.
- The format is spelled out (column 1 X, column 2 Y, column 3 Z, first row ignored) so the system knows exactly how to extract the data.
- A 3D sketch is explicitly requested to place the extracted points.



CAD Macros AI Generation

Common Examples

DXF export

Export all my part as DXF with multiple criteria and constraints

What a Bad Prompt Looks Like?

Export to DXF.

Why this prompt is bad?

- It doesn't say which document is open (a part? An assembly?)
- It doesn't say which parts should be exported.
- It gives no workflow details like unfolding or filtering sheet metal.
- It provides no export path or interface requirements.

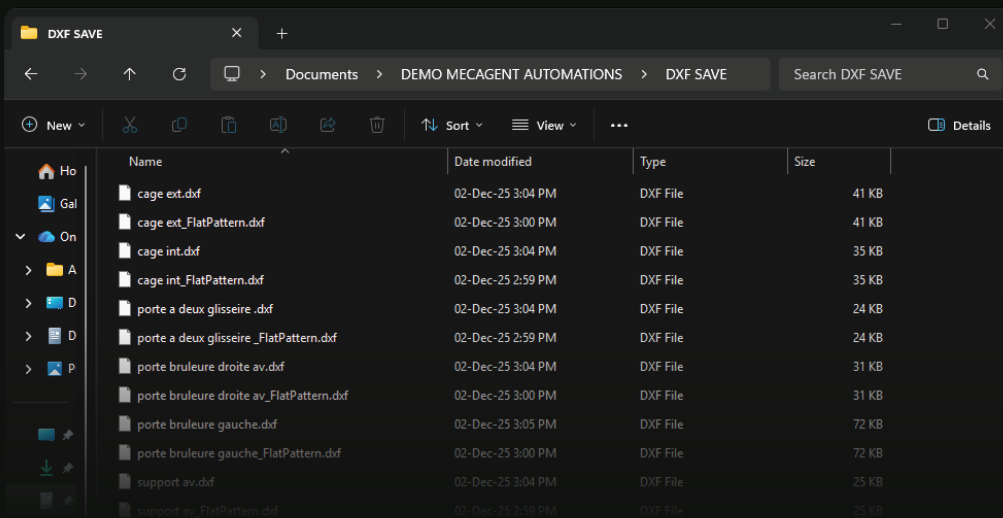
What does a good prompt looks like?

In my current assembly, create an automation that loops through all the parts and exports all my sheet metal part as DXF.

- Generate a small interface that lets me choose the export path.
- Unfold each part before exporting.
- Ignore non-sheet metal parts.

Why this prompt works?

- It specifies that an assembly is currently open, so MecAgent knows the document context.
- It clearly defines the target parts (only sheet metal parts) and tells the system to ignore all others.
- It instructs MecAgent to iterate through every part in the assembly, which sets the correct looping behavior.
- It requests a dialog interface for selecting the export folder, so the system knows how to gather user input.
- It explicitly describes the required preprocessing step (unfolding the part) before exporting.



CAD Macros AI Generation

Common Examples

Applying fillets

Applying fillets to a part with specific criteria. Here, to all the rectangular holes

What a Bad Prompt Looks Like?

Apply fillets on all the square holes

Why this prompt is bad?

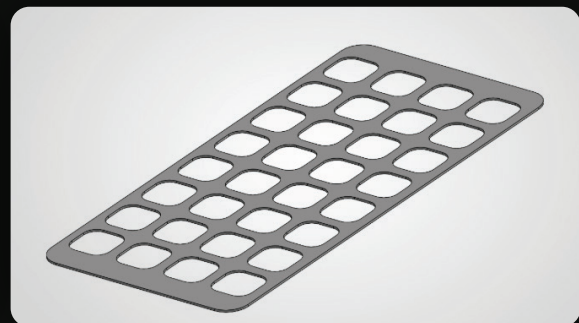
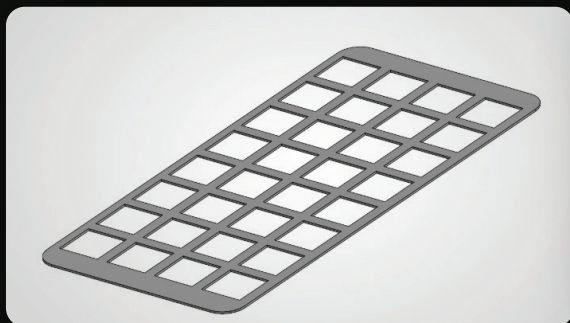
- It doesn't explain what "square holes" refers to in the model, and MecAgent has no way to guess which ones you mean.
- A square hole contains many edges, and the prompt doesn't define which of those edges should receive the fillet.
- It gives no selection rule, no geometry criteria, and no way for MecAgent to filter the correct edges.

What does a good prompt look like?

Create an automation that applies a fillet to the edges colinear with the X axis. Create a dialog on which I can select the fillet radius.

Why this prompt works?

- MecAgent needs a precise rule to know which edges should receive a fillet. By giving it a clear geometric criterion (edges colinear with the X axis), the system can filter the correct edges instead of applying fillets everywhere. This criterion is specific to this use case, but many others could be used depending on the situation, such as edge type (straight, circular, curved), direction, coordinates, minimum length, adjacency to a surface, and more.
- The prompt also asks for a dialog where the user can choose the fillet radius, which tells MecAgent how to collect the needed parameter before applying the feature.



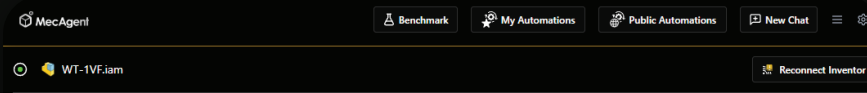
Key Feature

AI Mechanical Engineering Expert

MecAgent allows you to get mechanical engineering answers with AI directly in your workflow (design guidance, standards, calculations, and best practices), based on verified engineering sources.

AI MechE Expert Interface

MechE Sources Page



User
What is the maximum allowable deflection of a beam under load to meet functional alignment requirements?
05:23 PM

User's prompt input

Assistant
The maximum allowable deflection of a beam under load to meet functional alignment requirements is a serviceability limit defined to ensure that the beam's deflection does not impair the function, safety, or aesthetics of the structure or machine. This allowable deflection varies widely depending on the application, type of load, support conditions, and the sensitivity of the components or materials attached to or supported by the beam.

To copy-paste your prompt

Common engineering practice uses limits expressed as fractions of the beam span length L . Typical allowable deflection limits include:
- $\frac{L}{360}$: Used for many general structural applications where moderate deflection can be tolerated.
- $\frac{L}{1200}$: Applied in cases requiring exceptionally stiff girders or beams to maintain precise alignment.
- Very small limits such as 0.010 inch or less: Common for machine parts where maintaining functional alignment is critical, such as punch press columns [B].

To edit your prompt

The choice of deflection limit depends on the class of service, materials involved, and the potential damage excessive deflection might cause. For example, beams adjacent to brittle materials like plaster may have stricter limits (e.g. $\frac{L}{360}$) to prevent cracking [B].

To verify compliance with allowable deflection limits, the maximum deflection δ_{max} of a beam under a given load can be calculated using classical beam theory formulas. For example, for a simply supported beam with a central concentrated load F , the maximum deflection is given by:

$$\delta_{max} = \frac{FL^3}{48EI}$$

LLM explanation MechE question

where:
- F is the load,
- L is the span length,
- E is the Young's modulus of the beam material,
- I is the area moment of inertia of the beam's cross-section [B] [4].

The calculated deflection should be compared to the allowable deflection, for example,

$$\delta_{allow} =$$

Source 1
Marks Calculations For Machine Design.pdf - Brown, Thomas H (Mechanical Design Engineering Handbook).pdf
Page 63
Open Document

To open the source

In summary, there is no single maximum allowable deflection value; it must be selected based on functional requirements, materials, and service conditions. Standard engineering references provide guidelines and formulas to determine both the maximum deflection and suitable allowable limits to ensure functional alignment and serviceability [2] [3].

- Sources:
- 1 Marks Calculations For Machine Design.pdf - Brown, Thomas H (Mechanical Design Engineering Handbook).pdf (Page: 63)
 - 2 Machinery's Handbook Guide to Machinery's Handbook.pdf - Oberg, Erik, Jones, Franklin D., Horton, Holbrook L., Ryffel, Henry H. (Machinery's Handbook).pdf (Page: 548)
 - 3 Machinery's Handbook Guide to Machinery's Handbook.pdf - Oberg, Erik, Jones, Franklin D., Horton, Holbrook L., Ryffel, Henry H. (Machinery's Handbook).pdf (Page: 547)
 - 4 Marks Calculations For Machine Design.pdf - Brown, Thomas H (Mechanical Design Engineering Handbook).pdf (Page: 57)

05:23 PM

To write your prompt

MechE Sources
Ask me about any theoretical knowledge...
Automation MechE Sources Experimental

To send the prompt to the LLM

Change the feature

AI MechE Expert

How it works ?

Step by Step

1.

2.

3.

4.

5.

6.

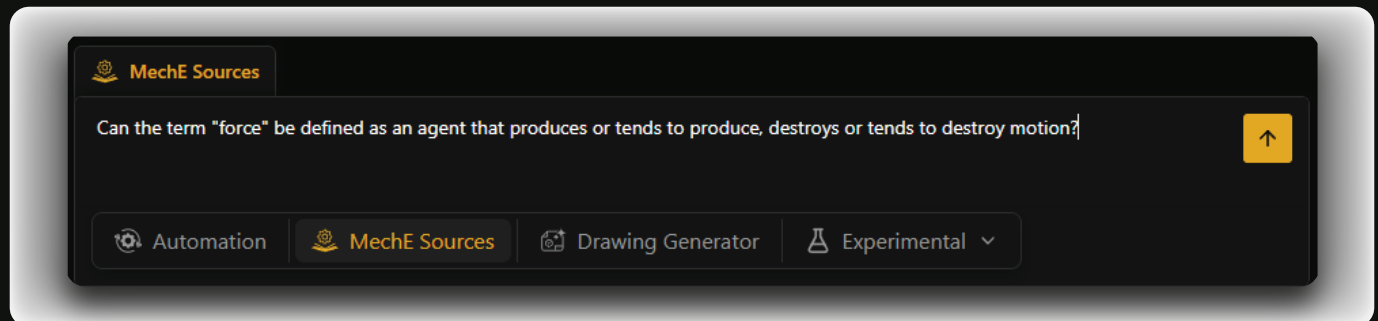
1. **You don't need to be connected** to the CAD Software to use it
2. Verify that you are in **"MechE Sources"** Mode
3. **Write whatever you want** until your request involves mechanical engineering knowledge
4. Press **Enter** or Click on the arrow button to send your request
5. The AI processes your request to answer your questions using the engineering books in our database. This takes between **30 seconds to 2 minutes**. Once ready, you'll hear a sound notification.
6. Click **"[1]"** to open the **source**.

Use this feature when you need validation, technical explanation, or engineering decision support during your design process.

AI MechE Expert

How it works ?

How to prompt



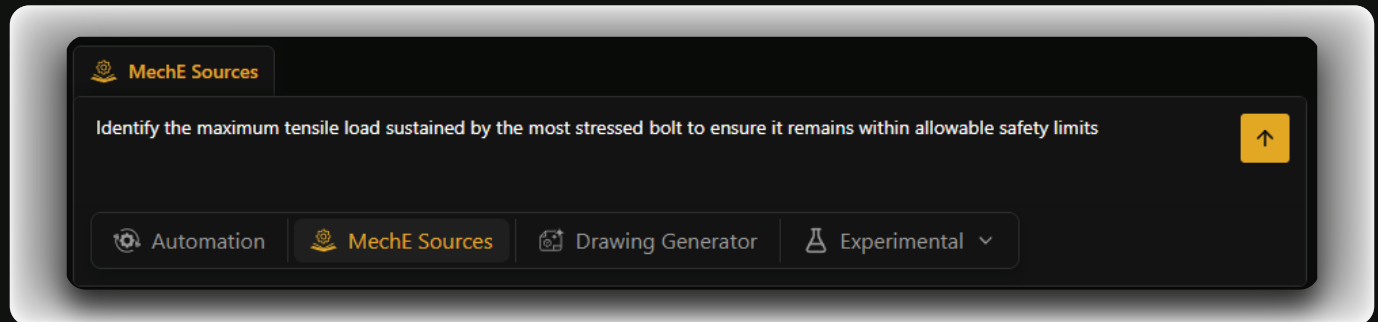
What you can ask

- ✓ "Can the term "force" be defined as an agent that produces or tends to produce, destroys or tends to destroy motion?"
--
- ✓ "Explain the difference between stress and strain."
--
- ✓ "What is the formula for tensile stress?"
--
- ✓ "How do I calculate the safety factor of a bolted connection?"
--
- ✓ "What are the main failure modes of a bolted joint?"
--
- ✓ "A steel roller with radius $R_1 = 18$ mm is in contact with a cam having a local equivalent radius $R_2 = 45$ mm. The maximum normal load is 6.5 kN, distributed over a contact width of 14 mm. Material properties:
 - Roller: $E_1 = 210$ GPa, $\nu_1 = 0.30$,
 - Cam: $E_2 = 210$ GPa, $\nu_2 = 0.30$. Allowable contact stress under cyclic loading: 1350 MPa.
 - Due to misalignment, the local load increases by 22%.
 - Question: "Does the maximum Hertzian contact pressure remain below the allowable limit? If not, determine the minimum required roller width to ensure safe operation."
--
- ✓ "What is the difference between static and fatigue loading?"

AI MechE Expert

How it works ?

How to prompt



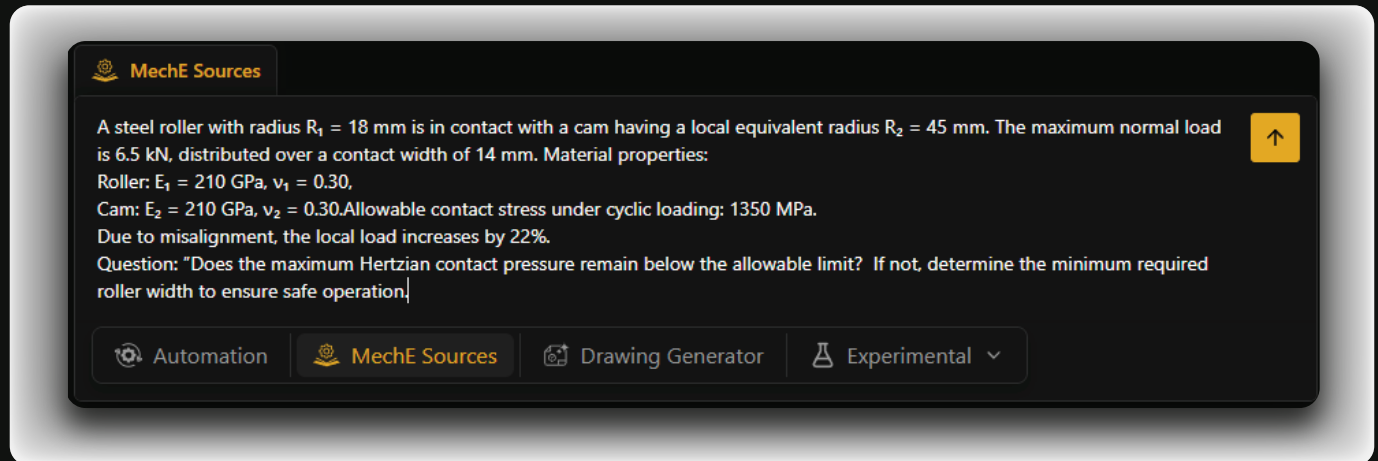
What you should not ask :

- ✘ "Identify the maximum tensile load sustained by the most stressed bolt to ensure it remains within allowable safety limits."
-
- ✘ "Confirm that this design is safe for production."
-
- ✘ "Certify that this part complies with aerospace/automotive safety requirements."
-
- ✘ "Validate the structural integrity of this assembly based only on partial information."
-
- ✘ "Determine whether this pressure vessel / lifting system / safety-critical component is compliant and safe to use."

Ask general mechanical engineering questions, get explanations of concepts, formulas, standards, and design principles. This feature is intended for engineering assistance and education, not for certified safety validation, final design approval, or safety-critical calculations without professional review.

AI MechE Expert

Advanced Prompting Techniques



Provide context

Specify the part, system, and use case.

Include numbers

Add dimensions, loads, materials, and conditions.

State your goal

Explain whether you want to understand, calculate, or compare.

Be specific

Focus on one clear question at a time.

Avoid validation questions

Ask for analysis, not certification.

Use proper units

Always include units: mm, MPa, kN, etc.

Break down complex problems

Ask step by step instead of asking everything at once.

Request assumptions and steps

Ask for formulas and reasoning when needed.

AI MechE Expert

Common Examples

Hertzian Contact Stress Verification (Roller-Cam System)

Prompt:

"How to calculate a pressure loss in a pipe ?"

MecAgent
Benchmark
My Automations
Public Automations
New Chat

XXXXXXXXXXXX
Reconnect SolidWorks

Hertzian Contact Stress Verification (Roller-Cam System)

Prompt:

"A steel roller with radius $R_1 = 18$ mm is in contact with a cam having a local equivalent radius $R_2 = 45$ mm. The maximum normal load is 6.5 kN, distributed over a contact width of 14 mm. Material properties: Roller: $E_1 = 210$ GPa, $\nu_1 = 0.30$, Cam: $E_2 = 210$ GPa, $\nu_2 = 0.30$.

Allowable contact stress under cyclic loading: 1350 MPa. Due to misalignment, the local load increases by 22%. Question: Does the maximum Hertzian contact pressure remain below the allowable? If not, determine the minimum required roller width to ensure safe operation.

05:04 PM

Assistant

The maximum Hertzian contact pressure for the roller-cam system under the increased load due to misalignment (22% increase) is calculated using the Hertzian contact stress formula for cylindrical contacts:

$$\sigma_{H \max} = \sqrt{\frac{F}{b} \cdot \left(\frac{1}{R_1} + \frac{1}{R_2}\right) \cdot \frac{\pi}{(1-\nu_1^2)/E_1 + (1-\nu_2^2)/E_2}}$$

where F is the effective normal load, b is the contact width, R_1 and R_2 are the radii of the roller and cam respectively, E_1 and ν_1 are the elastic modulus and Poisson's ratio of the materials.

Given:

- $R_1 = 18$ mm,
- $R_2 = 45$ mm,
- nominal load $F = 6.5$ kN,
- load increase factor due to misalignment = 1.22,
- contact width $b = 14$ mm,
- material properties $E_1 = E_2 = 210$ GPa, and $\nu_1 = \nu_2 = 0.30$,
- allowable Hertzian contact stress = 1350 MPa.

The effective load $F_{eff} = 6.5 \text{ kN} \times 1.22 = 7.93 \text{ kN}$.

Calculated maximum Hertzian contact stress is approximately 3996 MPa, which exceeds the allowable 1350 MPa, hence unsafe with current roller width.

To ensure safe operation, the minimum required roller width b_{min} must satisfy:

$$b_{min} = \frac{F_{eff} \times \pi \times \left(\frac{1}{R_1} + \frac{1}{R_2}\right)}{\sigma_{allowable} \times \left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2}\right)}$$

This yields a minimum roller width of approximately 123 mm.

Therefore, the roller width must be increased significantly from 14 mm to 123 mm to keep the Hertzian contact stress below the allowable limit under the given loading conditions.

use the classical Hertzian contact stress theory as documented in [1], [2], [3].

MechE Sources

Identify the maximum tensile load sustained by the most stressed bolt to ensure it remains within allowable safety limits.

Analysis and Design of Machine Elements

Solutions:

Steps	Computation	Results	Units
1. The number of cycles to failure N_f at each stress level σ_i	From Eq. (2.35)	$N_f = 0.01 \times 10^7$	
	$N_f = N_f \left(\frac{\sigma_i}{\sigma_s}\right)^{-m}$	$N_f = 0.0751 \times 10^7$	
	We have:	$N_f = 0.2497 \times 10^7$	
	$N_f = N_f \left(\frac{\sigma_i}{\sigma_s}\right)^{-m} = 10^7 \times \left(\frac{200}{500}\right)^{-10} = 0.01 \times 10^7$		
	$N_f = N_f \left(\frac{\sigma_i}{\sigma_s}\right)^{-m} = 10^7 \times \left(\frac{200}{500}\right)^{-10} = 0.0751 \times 10^7$		
	$N_f = N_f \left(\frac{\sigma_i}{\sigma_s}\right)^{-m} = 10^7 \times \left(\frac{200}{500}\right)^{-10} = 0.2497 \times 10^7$		
2. The number of remaining cycles N_r	From Miner's rule in Eq. (2.32)	$n_r = 4.21 \times 10^6$	
	$\frac{n_1}{N_1} + \frac{n_2}{N_2} + \dots = 1$		
	$4 \times \left(\frac{10^7}{0.0751 \times 10^7} + \frac{10^7}{0.2497 \times 10^7} + \frac{n_r}{n_r}\right) = 1$		
	Therefore:	$n_r = 0.2497 \times 10^7 \times \left(\frac{1}{1 - \frac{4 \times 10^7}{0.0751 \times 10^7 - 0.2497 \times 10^7}}\right) = 4.21 \times 10^6$	

2.4 Contact Strength

Previous discussions focus on strength analyses within an element to prevent body failure, such as yielding and fatigue fracture. The section deals with surface strength or contact strength in localized regions, with an aim to prevent surface failure.

2.4.1 Hertzian Contact Stresses

Contact is one of the most common methods of transmitting forces in a machine. When elements make contact with each other, a pair of equal and opposite forces generate according to the action-reaction law. Typical examples are the force transmission between a pair of meshing gears or rolling contact bearings. Theoretically, the contact between curved surfaces of elements is a point or a line. When curved elements are loaded, contact areas develop elastically from the basic surface curvatures. High contact stresses are correspondingly developed within small contact areas.

Contact stress (also called Hertzian contact stress) refers to the localized stress that develops as two curved surfaces come in contact and deform slightly under imposed loads. The magnitude of contact stress depends on the material properties, body geometries and applied loads [16]. Generally, small radii lead to small contact areas and, consequently, large contact stress.

Figure 2.8 illustrates the contact area and corresponding contact stress distribution within two cylinders with diameters of d_1 and d_2 , loaded with uniformly distributed force F along cylinder length l . The contact area is a narrow rectangle. The induced contact stress is three-dimensional but may be characterized by stress perpendicular to the plane of contact surface, with parabolic distribution, varying from the periphery of contact surface to the maximum at the centre [1]. Assume the contact is frictionless and the contacting bodies are elastic, isotropic and homogeneous. The maximum contact stress is calculated by the Hertz formula as [14]

$$\sigma_{\max} = \sqrt{\frac{F}{b} \times \left(\frac{1}{R_1} + \frac{1}{R_2}\right) \cdot \frac{\pi}{\left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2}\right)}} \quad (2.51)$$

where μ , E and ν are Poisson's ratio, elastic modulus and radius of curvature, respectively. The subscripts 1 and 2 refer to the two cylinders. The negative symbol "-" refers to an internal surface. The Hertz formula is the foundation for the calculation of load carrying capabilities of rolling contact bearings, gears and so on.

2.4.2 Surface Fatigue Failure

In machine elements like rolling contact bearings or mating gear teeth, contact stresses

Key Feature

CAD Drawing AI Generator

MecAgent allows you to generate manufacturing drawings with AI directly from your CAD models (automatic views, tables, annotations, and standard drawing elements). For the moment the drawing generation doesn't include the GD&T or complex or company-specific BOM structures

CAD Drawing AI Generator Interface

Active Part

The image shows the CAD Drawing AI Generator interface with several callouts pointing to specific features:

- Active Part Tab**: Points to the 'Active Part' tab in the top navigation bar.
- Batch Drawing Creation Tab**: Points to the 'Batch Drawing Creation' tab in the top navigation bar.
- Import your drawing template**: Points to the 'Drawing Template' dropdown menu.
- Change international standard**: Points to the 'International Standard' dropdown menu.
- Change Units**: Points to the 'Units' dropdown menu.
- Change sheet size**: Points to the 'Sheet Size' dropdown menu.
- Change view strategy**: Points to the 'View Strategy' dropdown menu.
- Swipe for multibody parts**: Points to the 'Multibody parts: one sheet per body' toggle switch.
- Drawing Generation Button**: Points to the 'Generate' button.
- Change the feature**: Points to the 'Drawing Generator' feature tab in the bottom navigation bar.

CAD Drawing AI Generator Interface

Batch Drawing Creation

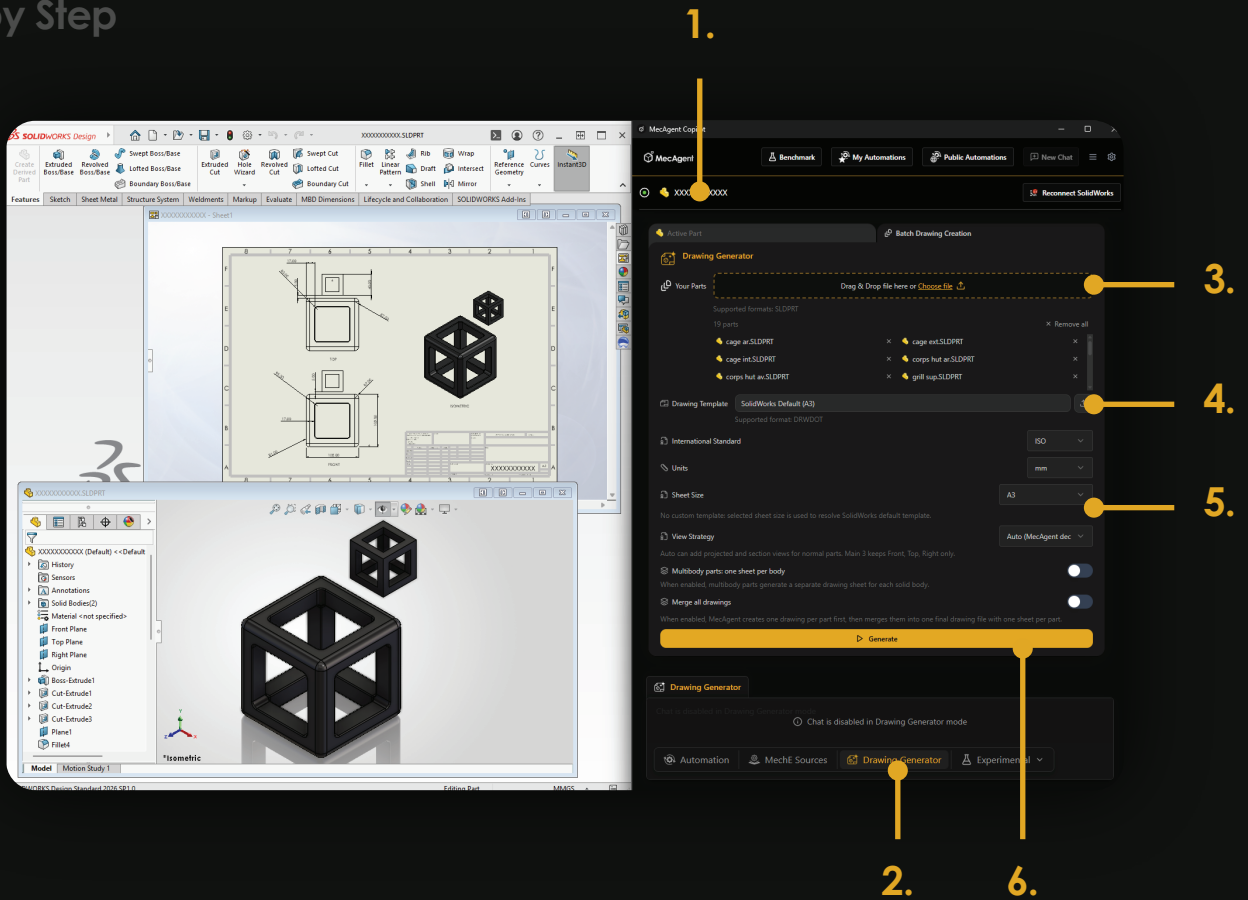
The screenshot shows the 'Batch Drawing Creation' interface. At the top, there are two tabs: 'Active Part' and 'Batch Drawing Creation'. The 'Batch Drawing Creation' tab is active and contains the following elements:

- Active Part Tab:** A label pointing to the 'Active Part' tab.
- Batch Drawing Creation Tab:** A label pointing to the 'Batch Drawing Creation' tab.
- Import your Parts:** A label pointing to the 'Your Parts' section, which includes a 'Drag & Drop file here or Choose file' button and a list of 19 parts (e.g., cage ar.SLDPRT, cage int.SLDPRT, corps hut ar.SLDPRT).
- List of imported parts:** A label pointing to the list of parts.
- Import your drawing template:** A label pointing to the 'Drawing Template' dropdown menu, currently set to 'SolidWorks Default (A3)'. Below it is the 'Supported format: DRWDOT'.
- Change international standard:** A label pointing to the 'International Standard' dropdown menu, currently set to 'ISO'.
- Change Units:** A label pointing to the 'Units' dropdown menu, currently set to 'mm'.
- Change sheet size:** A label pointing to the 'Sheet Size' dropdown menu, currently set to 'A3'.
- Change view strategy:** A label pointing to the 'View Strategy' dropdown menu, currently set to 'Auto (MecAgent dec)'. Below it are two toggle switches for 'Multibody parts: one sheet per body' and 'Merge all drawings', both currently turned on.
- Swipe for multibody parts:** A label pointing to the 'Multibody parts: one sheet per body' toggle switch.
- Merge all the drawings:** A label pointing to the 'Merge all drawings' toggle switch.
- Drawing Generation Button:** A label pointing to the 'Generate' button at the bottom of the interface.
- Change the feature:** A label pointing to the 'Drawing Generator' feature in the bottom navigation bar, which also includes 'Automation', 'MechE Sources', and 'Experimental'.

CAD Drawing AI Generator

How it works ?

Step by Step



1. Verify that you are **correctly connected** to the CAD software (if not, go to the section “Connection issues with the CAD software”).
2. Verify that you are in “**Drawing Generator**” mode.
3. Drag and **drop** your file **or choose your file** (if you are in the Active Part window, you can directly generate the drawing for your part).
4. If you have one, **import** your company drawing **template**.
5. If needed, choose your standard, units, sheet size, and the view strategy you want.
6. Press **Enter** or click on the **Generate** button to send your request.
7. The AI processes your request to create the drawing. This takes a few seconds. Once ready, you will hear a notification sound.
8. You can review your drawing to confirm that it meets your specifications and that all dimensions are correctly positioned.

CAD Drawing AI Generator

What drawings can it generate?

Drawings it can generate:

- ✓ "Generate standard drawing views (front, top, side, isometric)"
--
- ✓ "Add basic dimensions (linear, diameter, radius)"
--
- ✓ "Create section and detail views"
--
- ✓ "Adjust scale and layout of the drawing"
--
- ✓ "Place common annotations and standard drawing elements"
--
- ✓ "Organize views on the sheet automatically"

Drawings it can't generate:

- ✗ "Generate full GD&T annotations (position, flatness, profile, etc.)"
--
- ✗ "Create complex or company-specific BOM structures"
--
- ✗ "Produce fully production-ready drawings requiring strict standards compliance"
--
- ✗ "Handle highly customized drawing templates with advanced rules"
--
- ✗ "Guarantee that the drawing meets certification or regulatory requirements"

CAD Drawing AI Generator

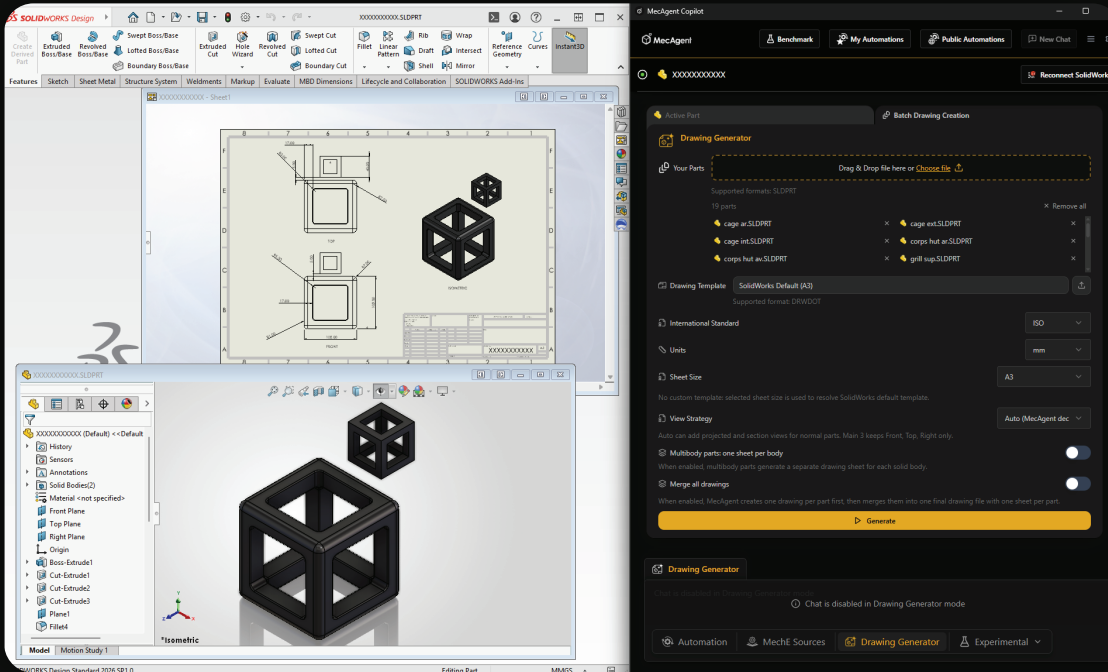
Technical Recommendations

- Drawing generation works on the currently active part or on individual parts within an assembly. It does not support generating drawings for a full assembly, including BOMs or exploded views.
- We recommend **importing** your company's **drawing template**.
- We recommend **fully dimensioning all part features** and, if possible, **defining tolerances directly on the 3D model**.

CAD Drawing AI Generator

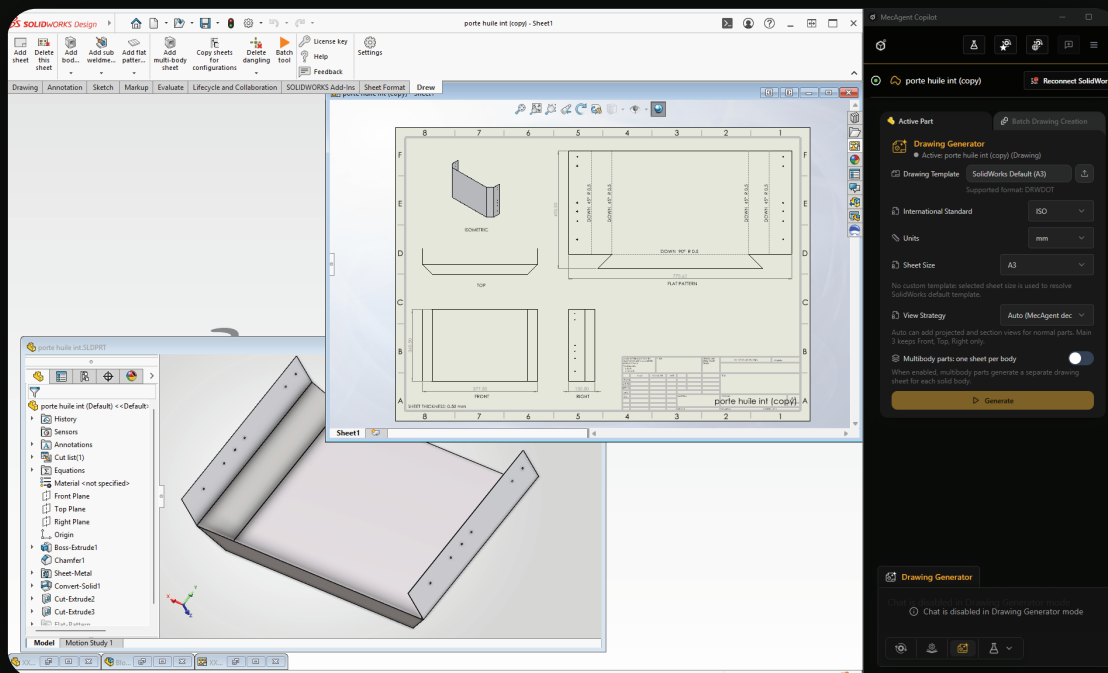
Common Examples

Classic Part



Sheet Metal Part

(<https://www.youtube.com/watch?v=8x9B-afD42l>)



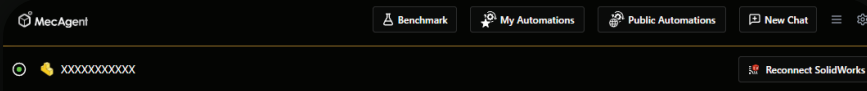
R&D Feature

AI CAD Copilot

MecAgent acts as an AI CAD copilot to perform actions directly in your CAD software (simple text-to-CAD, renaming parts and features, updating properties, applying materials and appearances, translations, and accessing specific metadata).

AI CAD Copilot Interface

CAD Copilot page



User
apply a shiny and expensive material to my part
10:30 AM

User's prompt input

list_available_materials completed
apply_material_to_part completed
get_current_materials completed

Assistant

Successfully applied **Chrome Stainless Steel** to your part! This premium material gives your part a shiny, polished, and expensive appearance. The material has been applied from the SOLIDWORKS Materials database and is now set on your solid body. Chrome Stainless Steel is known for its reflective, high-quality finish that looks both professional and luxurious.

10:31 AM

To copy-paste your prompt

To edit your prompt

User
ok now translate the name of my part and features in my feature tree from english to chinese
10:31 AM

get_textual_feature_tree completed
modify_feature_property_by_path completed
modify_feature_property_by_path completed
modify_feature_property_by_path completed
modify_feature_property_by_path completed
modify_feature_property_by_path completed
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modify_feature_property_by_path completed
modify_feature_property_by_path completed
modify_feature_property_by_path completed
get_textual_feature_tree completed

AI MCP Steps

Assistant

Successfully translated all feature names from English to Chinese!

Translation Summary:

Reference Planes: • Front Plane → 前基准面 (Front Reference Plane) • Top Plane → 顶基准面 (Top Reference Plane)

• Right Plane → 右基准面 (Right Reference Plane)

Sketches: • Sketch1 → 草图1 (Sketch 1) • Sketch7 → 草图7 (Sketch 7) • Sketch8 → 草图8 (Sketch 8)

Features: • Boss-Extrude1 → 凸台-拉伸1 (Boss-Extrude 1) • Chamfer1 → 倒角1 (Chamfer 1) • Sheet-Metal2 → 钣金2 (Sheet Metal 2) • Convert-Solid1 → 转换为钣金1 (Convert to Sheet Metal 1) • Cut-Extrude2 → 切除-拉伸2 (Cut-Extrude 2) • Cut-Extrude3 → 切除-拉伸3 (Cut-Extrude 3) • Flat-Pattern2 → 展开图样2 (Flat Pattern 2)

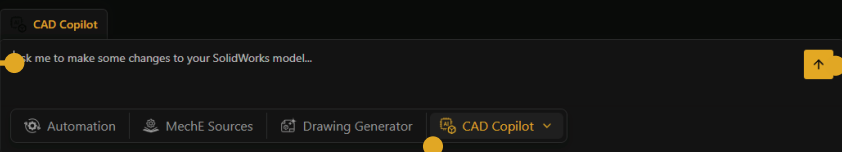
All 13 features have been successfully renamed to their Chinese equivalents. Your feature tree is now fully localized in Chinese! cn

10:32 AM

LLM explanation MechE question

To copy-past the answer

To write your prompt



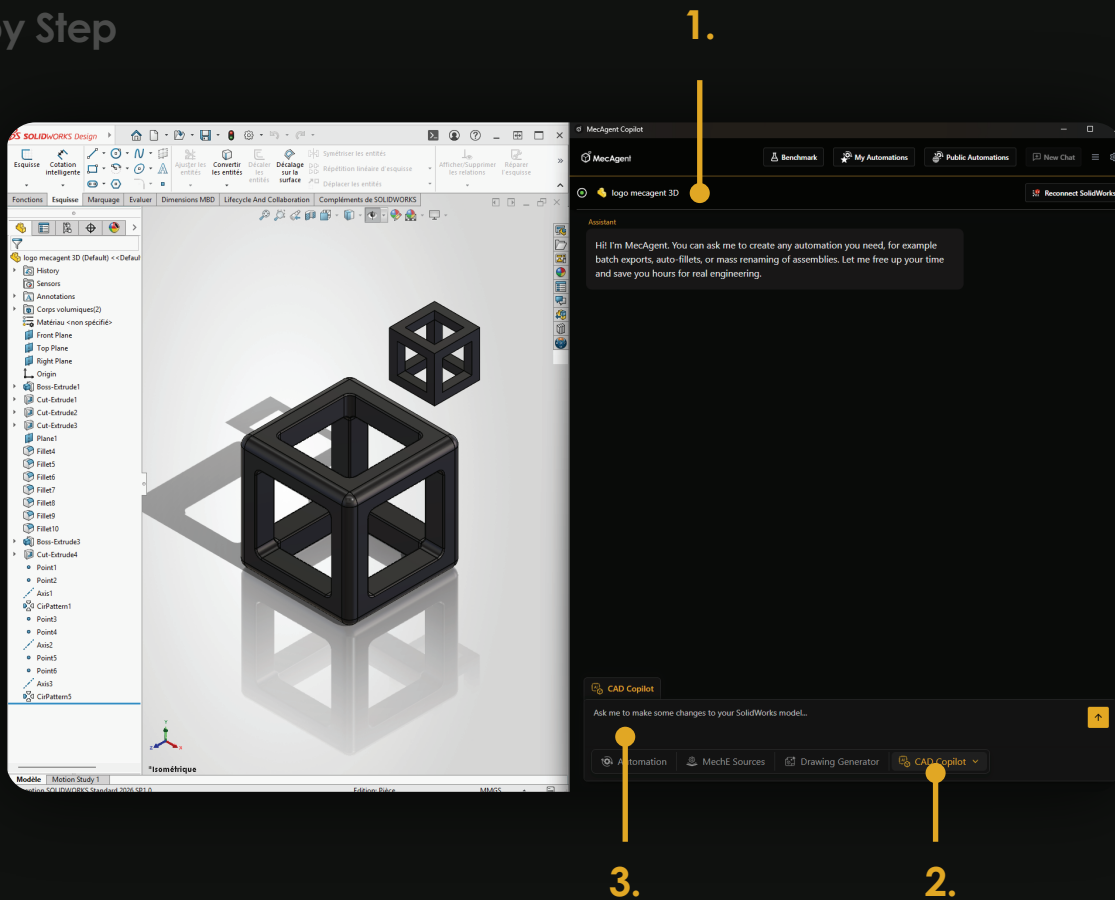
To send the prompt to the LLM

Change the feature

AI CAD Copilot

How it works ?

Step by Step



1. Verify that you are **correctly connected** to the CAD software (if not, go to the section “**Connection issues with the CAD software**”).
2. Verify that you are in “**CAD Copilot**” mode.
3. Write whatever you want to do on your part

Today, CAD Copilot can perform the following tasks within CAD software:

Use features such as: 2D Sketch, Extrude, Revolve, Holes, Fillets, and Chamfers

Apply or change the material of a part, a group of parts within an assembly, or all parts in an assembly

Apply or modify the properties of a part or an assembly

Save or “Save As” a part, a group of parts within an assembly, or an entire assembly in any file format with flexible or specific naming conventions

This functionality enables Text-to-CAD workflows, generating a feature tree and a modifiable part within CAD software. However, at present, the parts that can be created remain very limited and simple, as they can only be built using basic features (2D Sketch, Extrude, Revolve, Holes, Fillets, Chamfers), with limited context and spatial understanding.

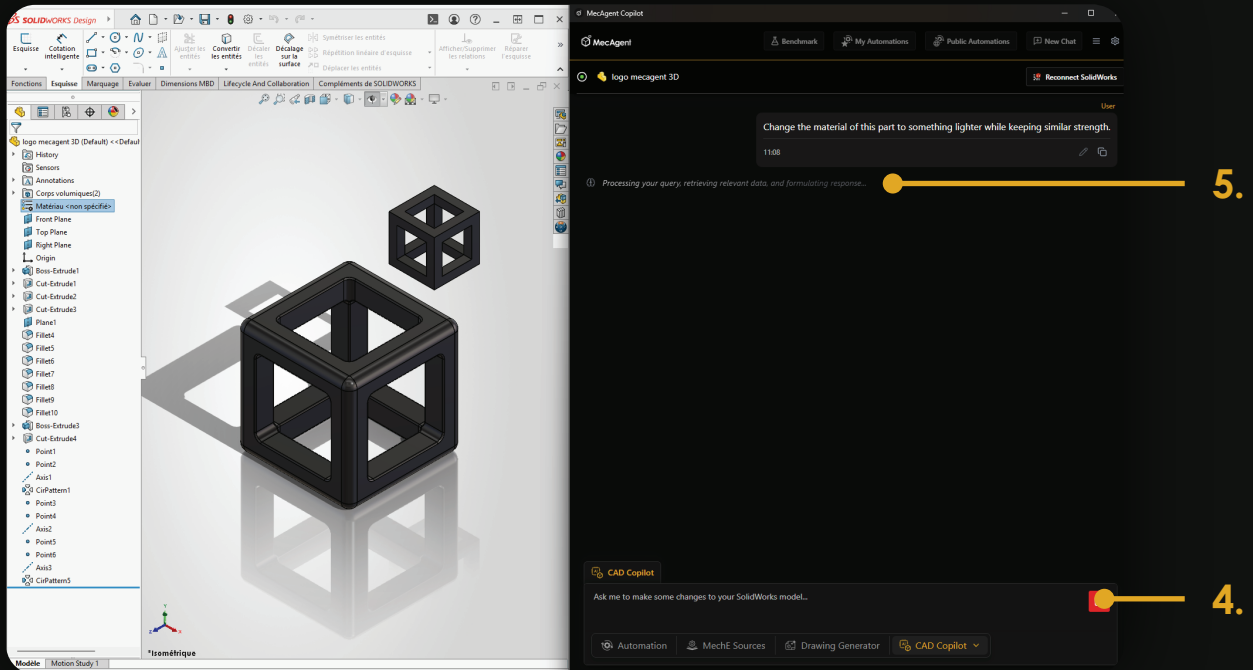
The Copilot has a notion of what the part is and maintains a snapshot view of the part or assembly at a given moment, but its contextual awareness and spatial understanding remain limited.

The Copilot can identify which face(s), edge(s), or point(s) are selected and can use this information to apply actions within the software.

AI CAD Copilot

How it works ?

Step by Step



4. Press **Enter** or click on the **Generate** button to send your request.
5. The CAD Copilot processes your request to create/apply the task you ask. This takes a few minutes. Once ready, you'll hear a sound notification.

It is an AI system equipped with tools that allow it to interact with and use specific functionalities of CAD software.

Today, the CAD Copilot can perform the following tasks within the CAD environment:

- Use core features: 2D Sketch, Extrude, Revolve, Holes, Fillets, Chamfers
- Apply or change materials on a part, a group of parts within an assembly, or an entire assembly
- Apply or modify properties of a part or an assembly
- Save or save as a part, a group of parts within an assembly, or a full assembly, in any file format with flexible naming conventions

AI CAD Copilot

How it works ?

Capabilities & limitations

This enables **Text-to-CAD workflows** with a **feature tree** and **fully editable parts** directly in **CAD software**. However, the parts that can currently be generated are limited to **simple geometries**, built only using the **supported features** listed above, with a **constrained understanding of spatial context**.

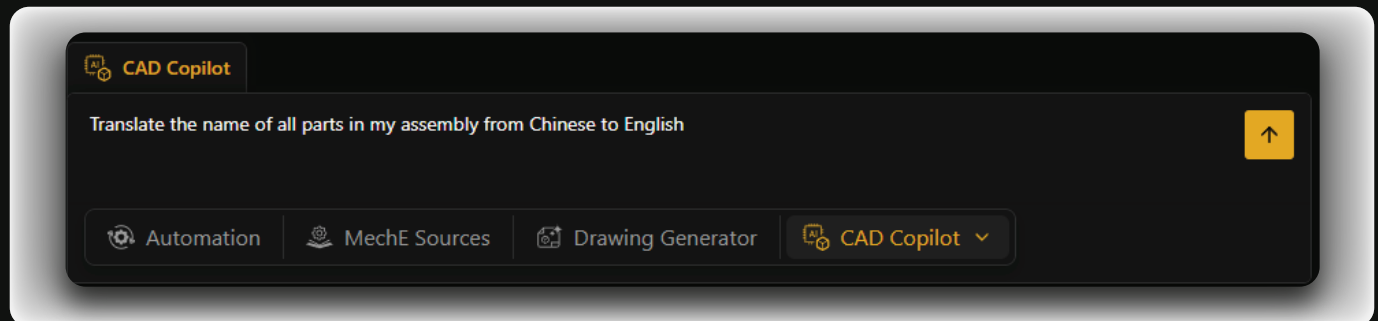
The copilot maintains a **partial understanding** of the part or assembly at a given moment, but its overall **spatial awareness** and **context** remain **limited**. It can **identify selected faces, edges, and points**, and use them to perform actions within the software.

This feature is **experimental**. Our R&D team is actively improving it, and many operations are still slower than performing them manually. It is made available to users to explore its potential and provide early value.

AI CAD Copilot

How it works ?

How to prompt



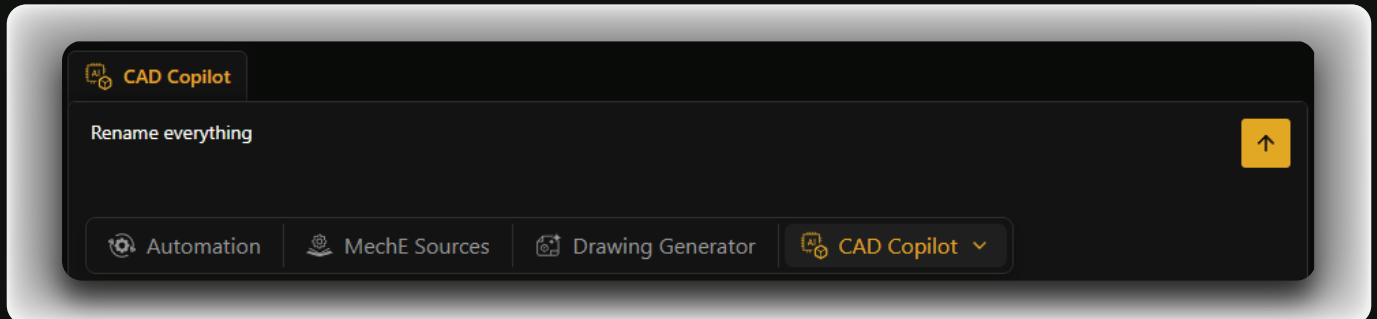
What you can ask

- ✓ "Translate the name of all parts in my assembly from Chinese to English"
--
- ✓ "Rename selected parts or features with a clear pattern"
--
- ✓ "Update custom properties (material, description, metadata)"
--
- ✓ "Apply materials and appearances to selected components"
--
- ✓ "Create simple parametric parts with defined dimensions (e.g. flange, plate, basic features)"
--
- ✓ "Add simple features (fillet, chamfer, hole) on selected geometry"
--
- ✓ "Access and extract specific metadata from parts or assemblies"

AI CAD Copilot

How it works ?

How to prompt



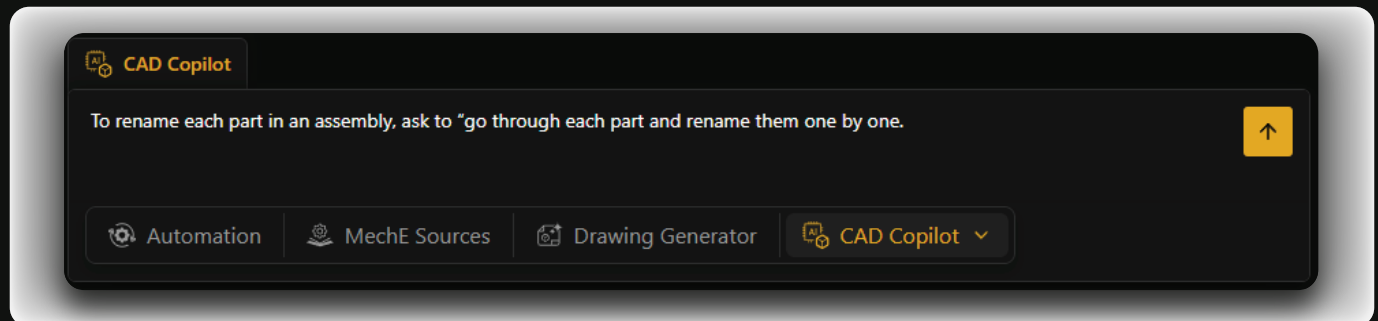
What you should not ask :

- ✗ "Rename everything without clear selection or rules"
--
- ✗ "Create complex objects with vague or high-level intent (e.g. "create a car")"
--
- ✗ "Generate advanced geometry requiring complex surfacing (lofts, complex surfaces)"
--
- ✗ "Perform highly customized or company-specific CAD workflows with strict rules"
--
- ✗ "Execute ambiguous actions without clear inputs, selections, or parameters"

AI CAD Copilot

Advanced Prompting Techniques

Ask for simple tasks in a structured way, such as changing part names or materials.



Part Creation:

Go step by step:

"Create a 500 × 200 × 2 mm plate" → "Add 5 mm diameter holes 10 mm from the edges at each corner of the plate" → "Add a 50 mm diameter hole at the center of the plate"

Rather than: "Create a 500 × 200 × 2 mm plate with 5 mm diameter holes 10 mm from the edges at each corner and a 50 mm diameter hole at the center of the plate" in a single prompt.

Translation:

To rename each part in an assembly, ask to "go through each part and rename them one by one."

CAD Part Creation / Modification:

When creating a part, specify as many dimensions as possible. For example, instead of saying "add holes in the corners," indicate the exact distance from the edges. Refer to existing elements on the 3D model and include as much precise, quantitative information as possible in your prompt (distances, axes, faces, etc.).

Apply / Change Material:

You can specify texture, color, and density, or clearly describe the type of material you need. For example: "I need a material that resists moisture over time and is suitable for plastic injection molding."

Apply / Change feature name:

The software allows renaming features, a functionality that is rarely used in SolidWorks due to time constraints. However, you can ask the copilot to rename each feature in the feature tree with consistent and meaningful names, based on the geometry created by each feature.

AI CAD Copilot

Common Examples

Translation

Demo: <https://youtu.be/sLhCmpzmFvE?si=J00ED7SiVXTiPPTw>

“Translate the name of all parts in my assembly from English to Russian and keep the original naming structure.”

The image shows a SolidWorks CAD environment with a gear assembly model. The MecAgent AI Copilot interface is overlaid on the right side. The chat window contains the following text:

User: Translate the name of all parts in my assembly from English to Russian and keep the original naming structure.

Assistant: Hi! I'm MecAgent. You can ask me to create any automation you need, for example batch exports, auto-fillets, or mass renaming of assemblies. Let me free up your time and save you hours for real engineering.

User: Translate the name of all parts in my assembly from English to Russian and keep the original naming structure.

Assistant: 13:53

- ✓ list_components_for_renaming completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
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- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed
- ✓ rename_part_in_assembly completed

Assistant: Translated 11 out of 14 unique parts from English to Russian. The assembly and all references have been updated.

✓ Successfully renamed:

- Shaft-1 → Вал-1
- Gearbox → Коробка передач
- 3-Speed Gear → Шестерня 3-й передачи
- CAD Copilot 3T → Шестерня 2-й передачи

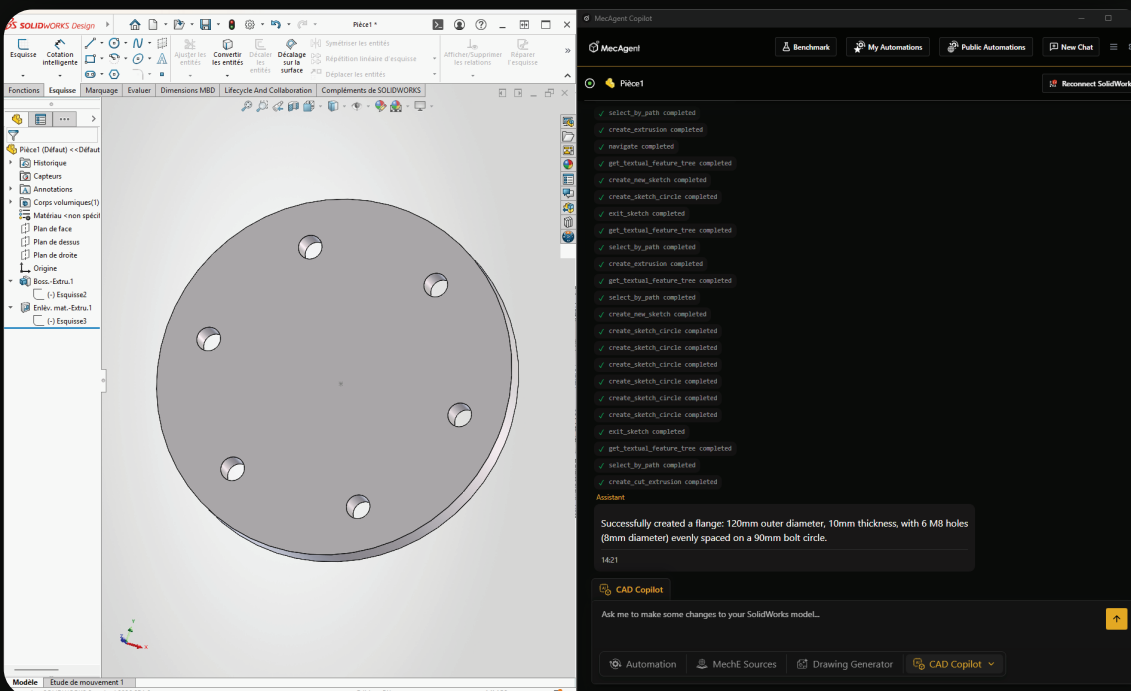
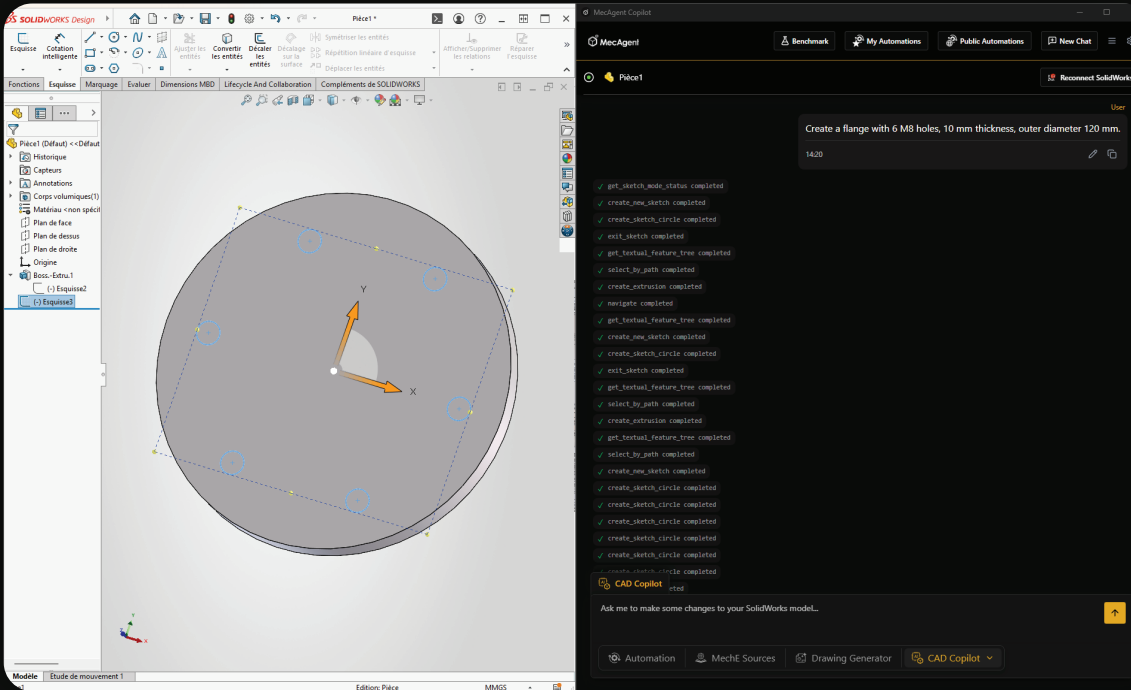
Ask me to make some changes to your SolidWorks model...

AI CAD Copilot Common Examples

Part Creation

Demo : https://youtu.be/8gz5TFuE_UI?si=k-YZLH8M07xsnSif

“Create a flange with 6 M8 holes, 10 mm thickness, outer diameter 120 mm.”

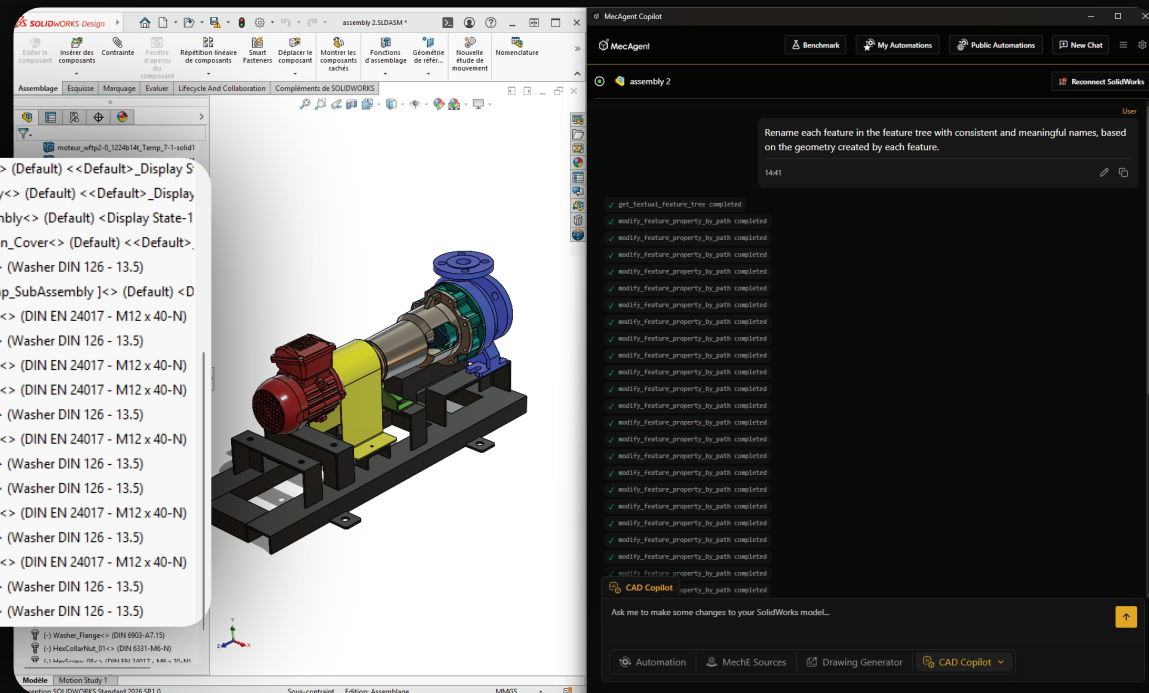
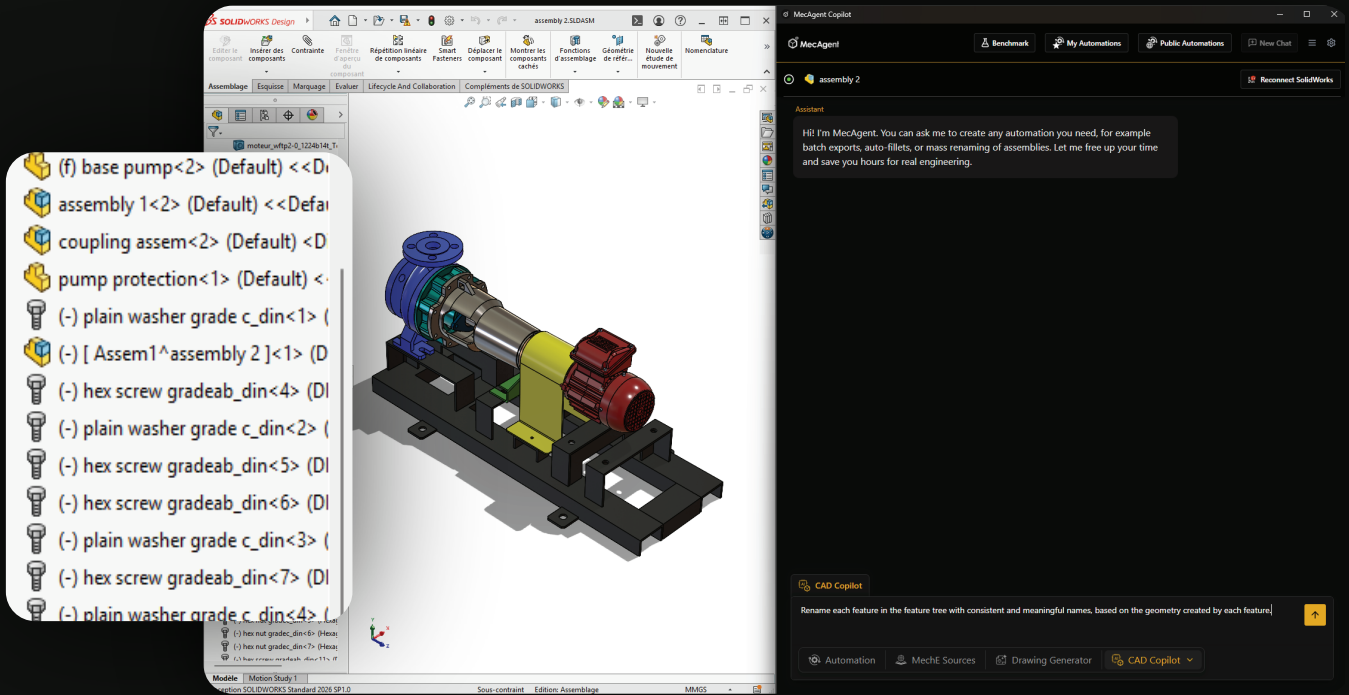


AI CAD Copilot

Common Examples

Apply / Change feature name

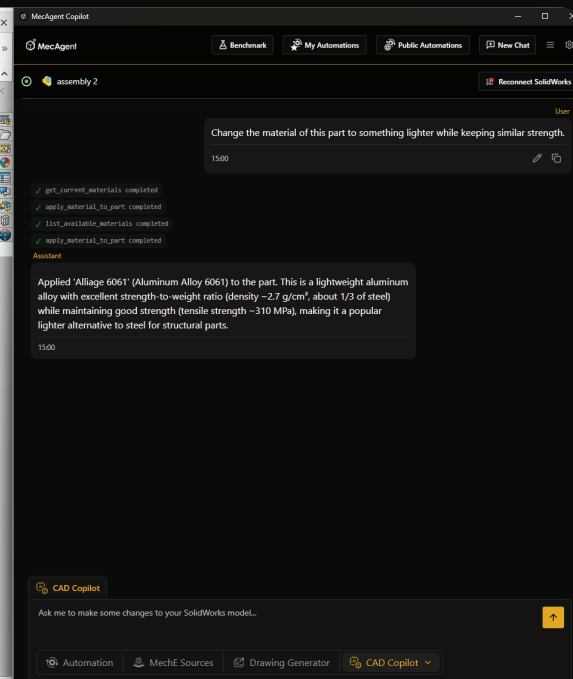
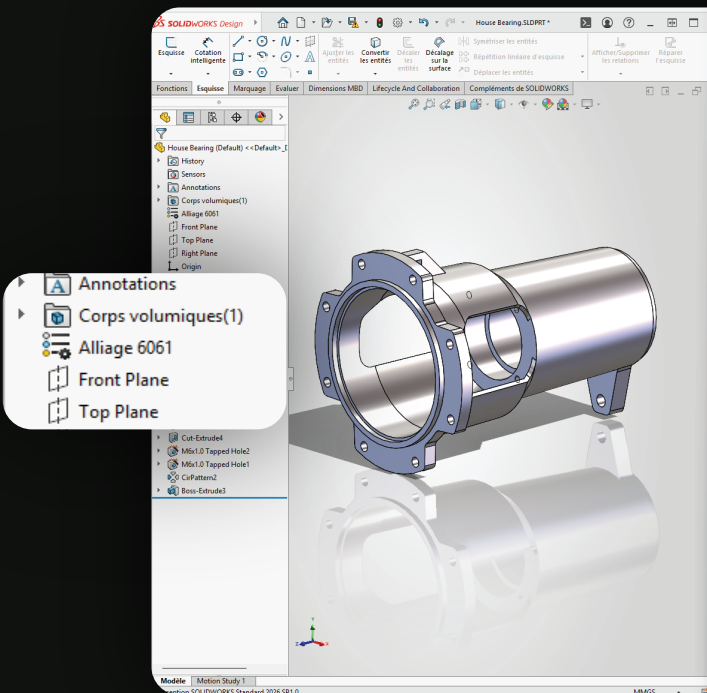
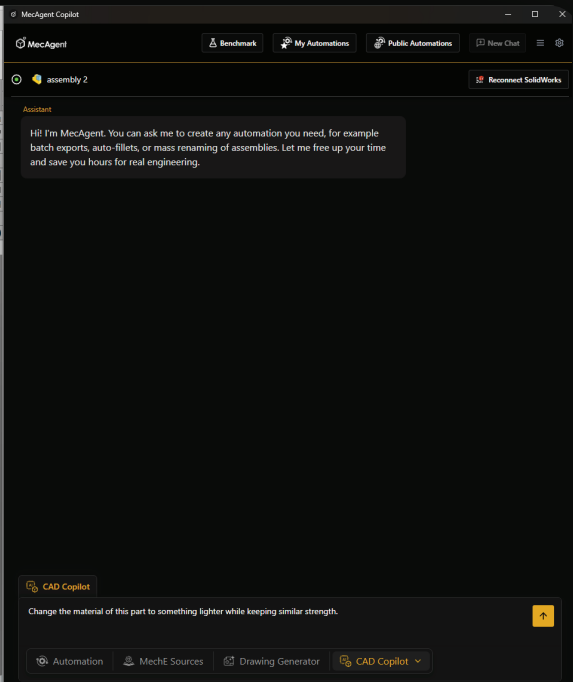
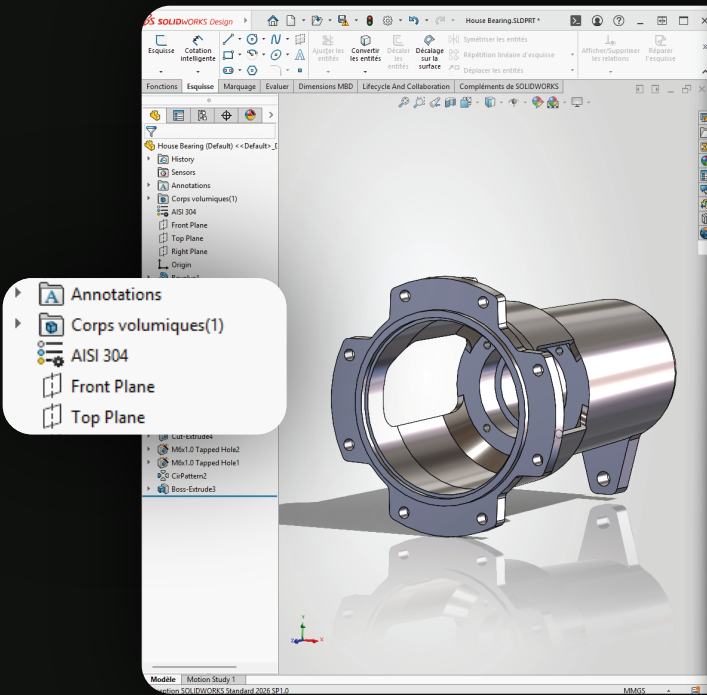
“Rename each feature in the feature tree with consistent and meaningful names, based on the geometry created by each feature.”



AI CAD Copilot Common Examples

Apply / Change Material

“Change the material of this part to something lighter while keeping similar strength.”



R&D Feature

Text to STL/STEP

MecAgent allows you to generate 3D parts from text as STEP or STL files, more complex geometries than the CAD copilot, no feature tree, ideal for standard parts like: screws, gears, ball bearings, and spacers...

Text to STL/STEP Interface

Text to STL/STEP page

The screenshot shows the MecAgent interface for generating CAD parts from text prompts. The interface includes a top navigation bar with 'Benchmark', 'My Automations', 'Public Automations', and 'New Chat'. Below this is a chat window where a user has entered a detailed prompt for a circular mechanical shaft with ten sections of varying diameters and lengths. The assistant has responded with a 'Part generated successfully' message, showing a 3D visualization of the shaft and a download button for 'stepped_shaft.stl'. The user has then entered a second prompt: 'Add 2 mm chamfers to each end'. The assistant has responded with another 'Part generated successfully' message, showing a 3D visualization of the shaft with chamfers and a download button for 'stepped_shaft_chamfered.stl'. The interface also features a bottom navigation bar with 'Automation', 'MechE Sources', 'Drawing Generator', and 'Text to STL/STEP'.

User's prompt input

To copy-paste your prompt

To edit your prompt

CAD part visualization

Choose an export format

Save your CAD Part in your folder

To write your prompt

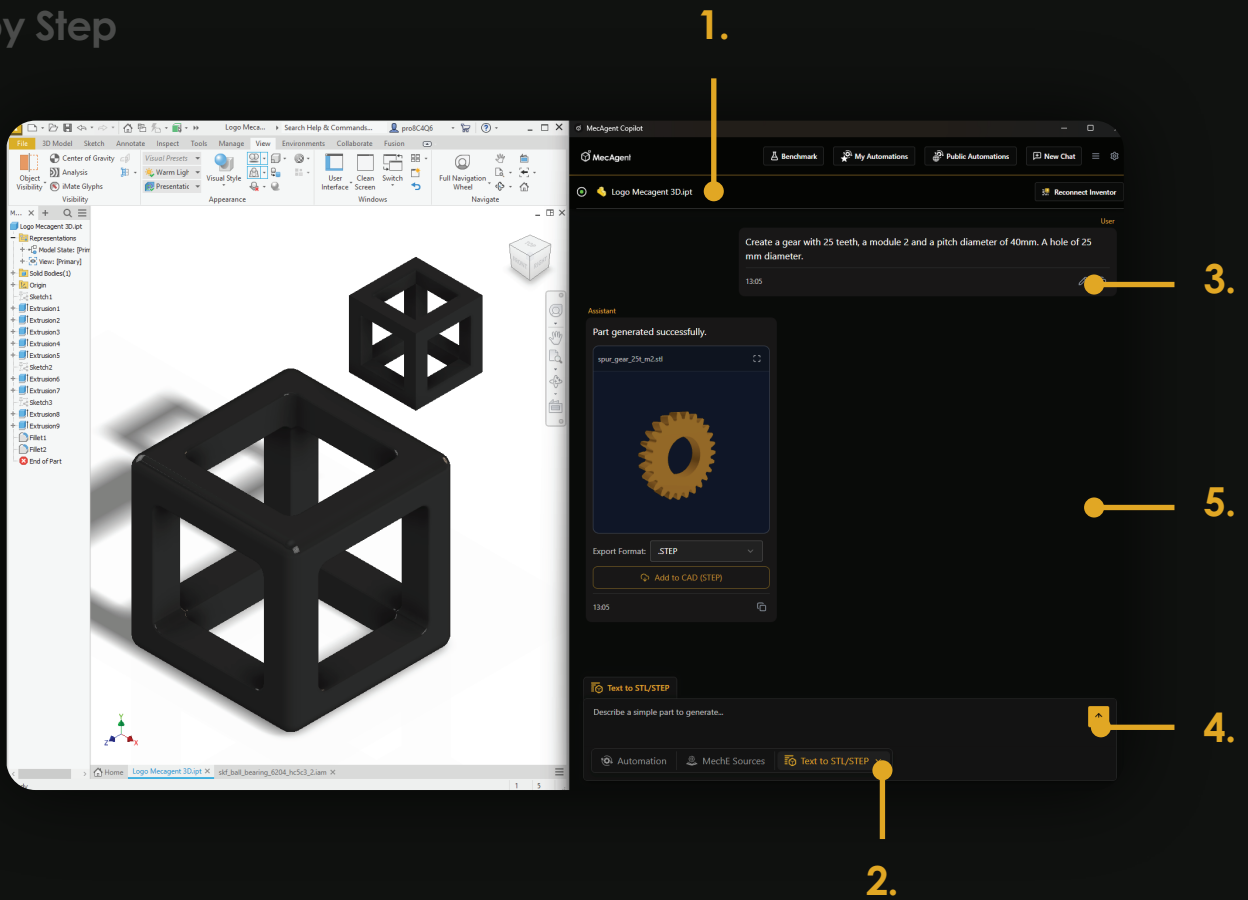
To send the prompt to the LLM

Change the feature

Text to STL/STEP

How it works ?

Step by Step

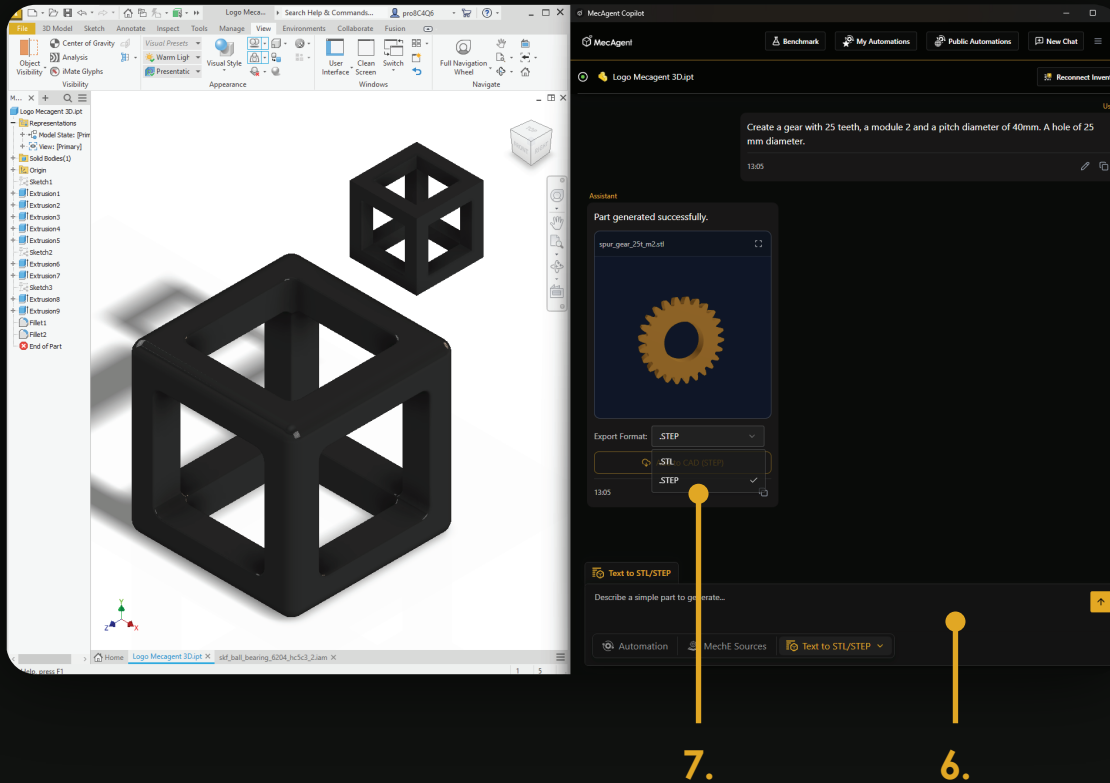


1. You don't need to be connected to the CAD Software to use it
2. Verify that you are in "Text to STL/STEP" mode.
3. Write the part you want to modelize with the description.
4. Press Enter or Click on the arrow button to send your request.
5. The AI processes your request to create the part. This takes a few minutes. Once ready, you'll hear a sound notification.

Text to STL/STEP

How it works ?

Step by Step



6. You can ask the AI to modify your part
7. Choose your export format between .STEP or .STL and click "Add to CAD" to save your file.
8. Save it in a specific folder
9. Open the part in a new file or Import the part in your assembly (we can't import the part without saving it in a specific folder)

Please note that it only creates non-editable parts as a single unified body, which can only be exported in STEP or STL format.

Text to STL/STEP

How it works ?

How to prompt



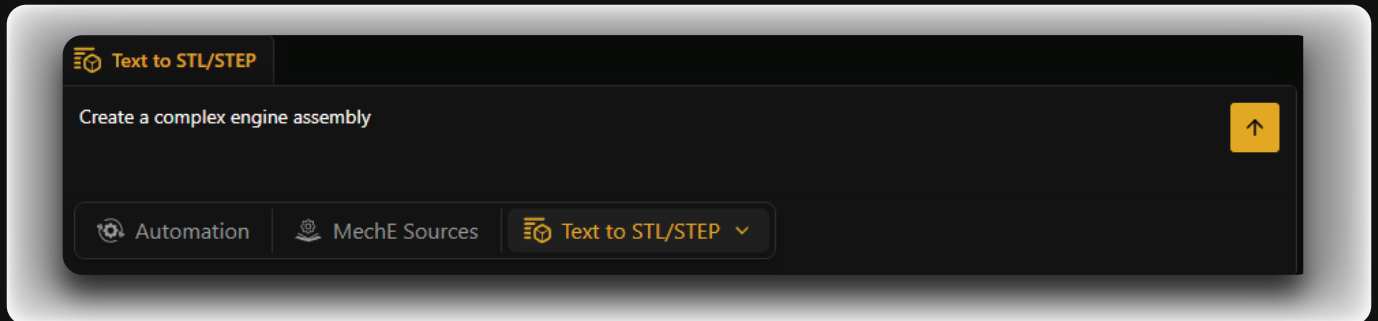
What you can ask

- ✓ "Create a bevel gear with 20 teeth, module 2, pressure angle 20°, and a pitch diameter of 40 mm"
--
- ✓ "Generate a standard hex bolt with M10 thread and 50 mm length"
--
- ✓ "Create a ball bearing with inner diameter 20 mm and outer diameter 47 mm"
--
- ✓ "Generate a spacer with 10 mm inner diameter, 20 mm outer diameter, and 5 mm thickness"
--
- ✓ "Create mechanical parts with precise dimensions and parameters"
--
- ✓ "Generate moderately complex standalone geometries"

Text to STL/STEP

How it works ?

How to prompt



What you should not ask :

- ✘ "Create a complex engine assembly"
-
- ✘ "Generate multi-part assemblies with interactions between components"
-
- ✘ "Expect a parametric feature tree or editable CAD history"
-
- ✘ "Produce highly detailed, fully engineered systems with constraints and relations"
-
- ✘ "Create designs requiring advanced engineering validation or optimization"

Text to STL/STEP

Technical Recommendations

- describe **step by step**
- Describe with **dimensions**
- The system is optimized for generating geometric parts based on **standard primitives** (e.g., boxes and cylinders), rather than complex surface geometry.

Text to STL/STEP

Common Examples

Circular mechanical shaft

Demo : <https://youtu.be/-ZXk4wnjyEM?si=fMMbdceUOHkr27ua>

Prompt:

Step 1 :

"Create a circular mechanical shaft aligned along the Z axis, composed of multiple cylindrical sections.

Specifications (from bottom to top):

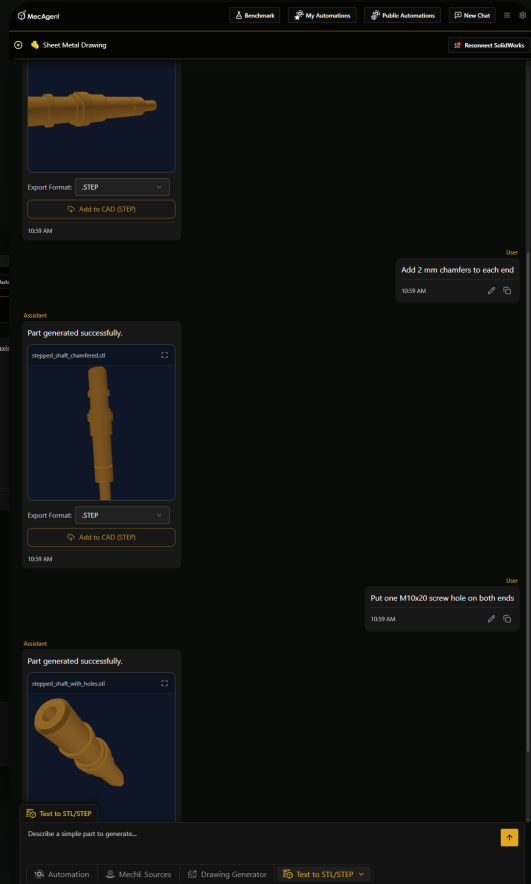
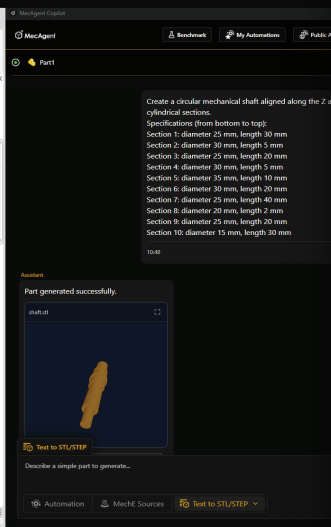
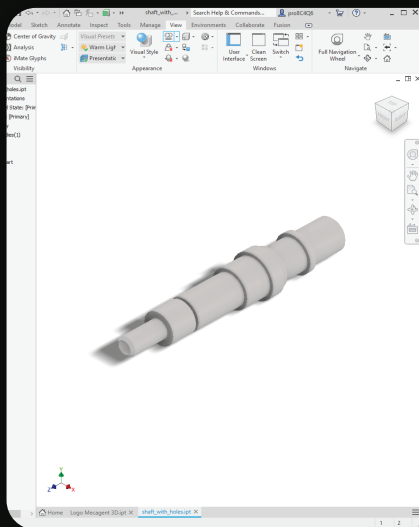
- Section 1: diameter 25 mm, length 30 mm
- Section 2: diameter 30 mm, length 5 mm
- Section 3: diameter 25 mm, length 20 mm
- Section 4: diameter 30 mm, length 5 mm
- Section 5: diameter 35 mm, length 10 mm
- Section 6: diameter 30 mm, length 20 mm
- Section 7: diameter 25 mm, length 40 mm
- Section 8: diameter 20 mm, length 2 mm
- Section 9: diameter 25 mm, length 20 mm
- Section 10: diameter 15 mm, length 30 mm"

Step 2 :

"Add 2 mm chamfers to each end"

Step 3 :

"Put one M10x20 screw hole on both ends"



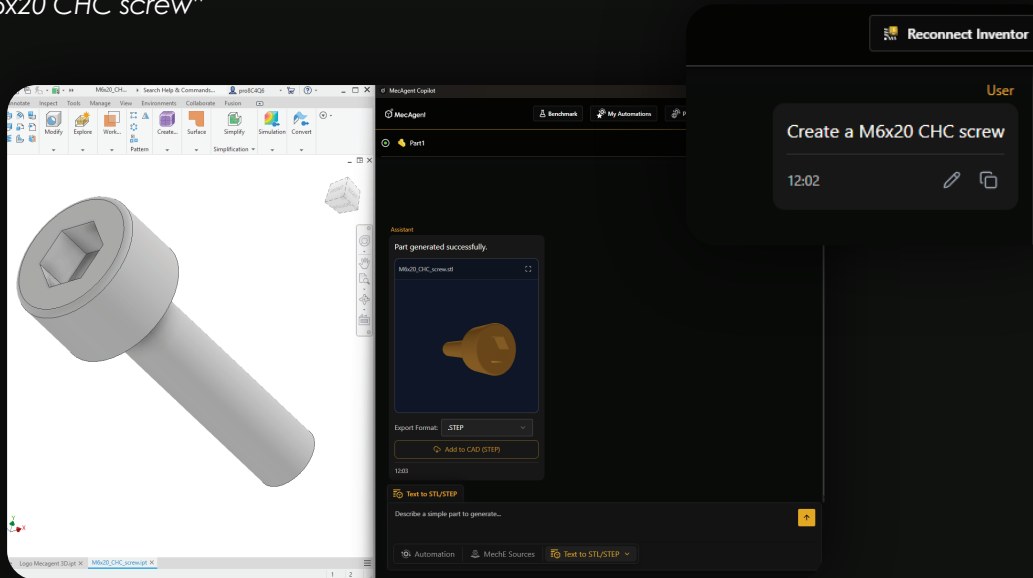
Text to STL/STEP

Common Examples

M6 screw

Prompt:

“Create a M6x20 CHC screw”

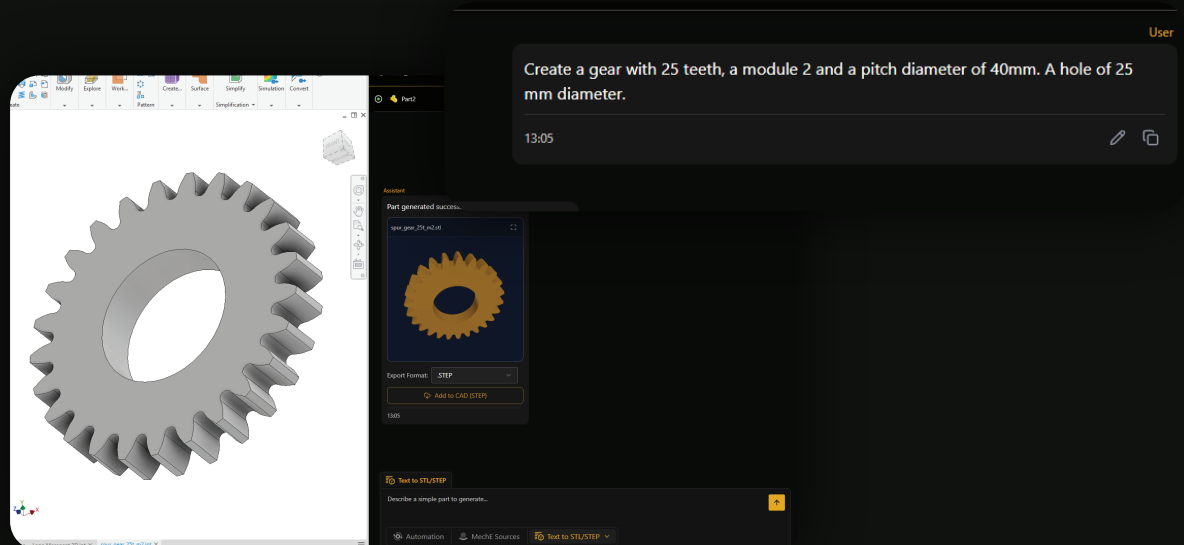


Gear

Demo : <https://youtu.be/SfW98fjEmzE?si=68UWrAuQmND3G1O2>

Prompt:

“Create a gear with 25 teeth, a module 2 and a pitch diameter of 40mm. A hole of 25 mm diameter.”



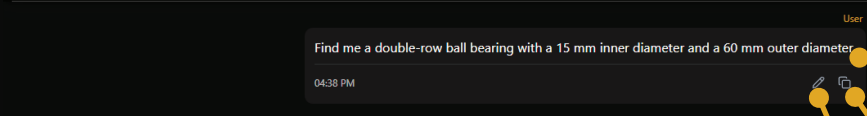
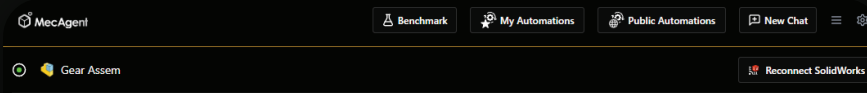
R&D Feature

Standard Part AI Finder

MecAgent allows you to find standard catalog parts using AI from natural language descriptions (identify components based on function or specifications and retrieve relevant parts).

Standard Part AI Finder Interface

Online Part Finder page



User's prompt input

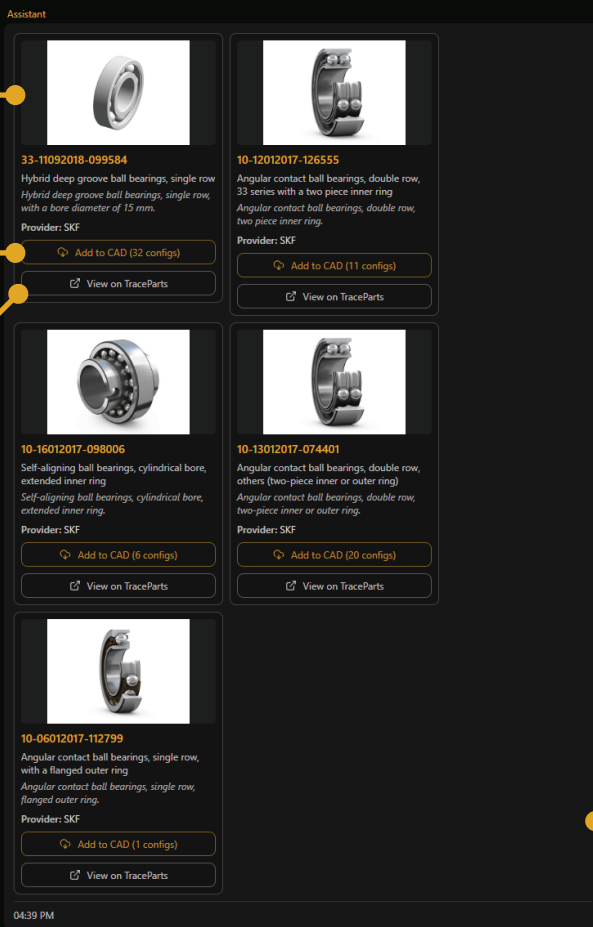
To copy-paste your prompt

To edit your prompt

Standard part visual

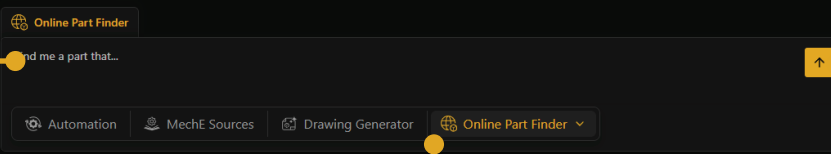
Access to all configurations

View the online ref.



Top database results

To write your prompt



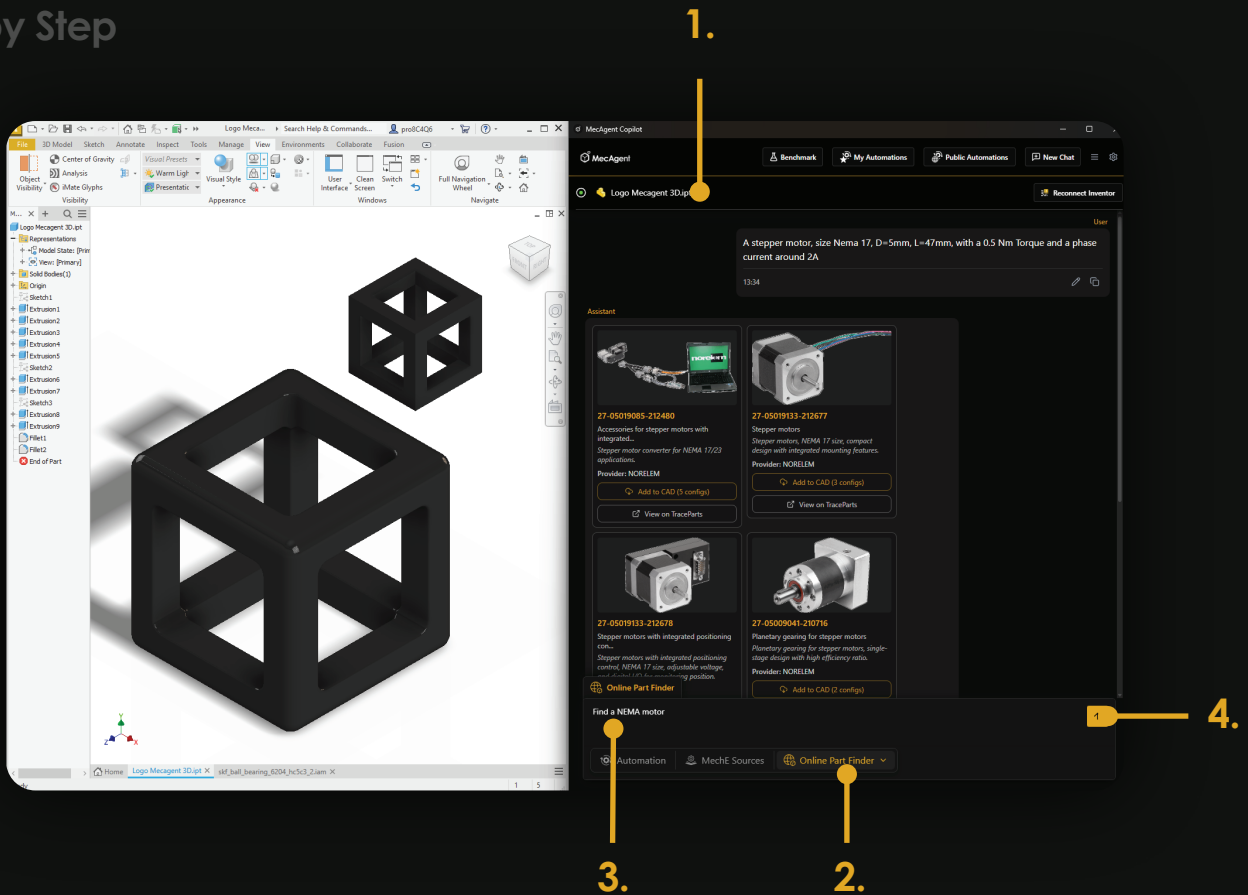
To send the prompt to the LLM

Change the feature

Standard Part AI Finder

How it works ?

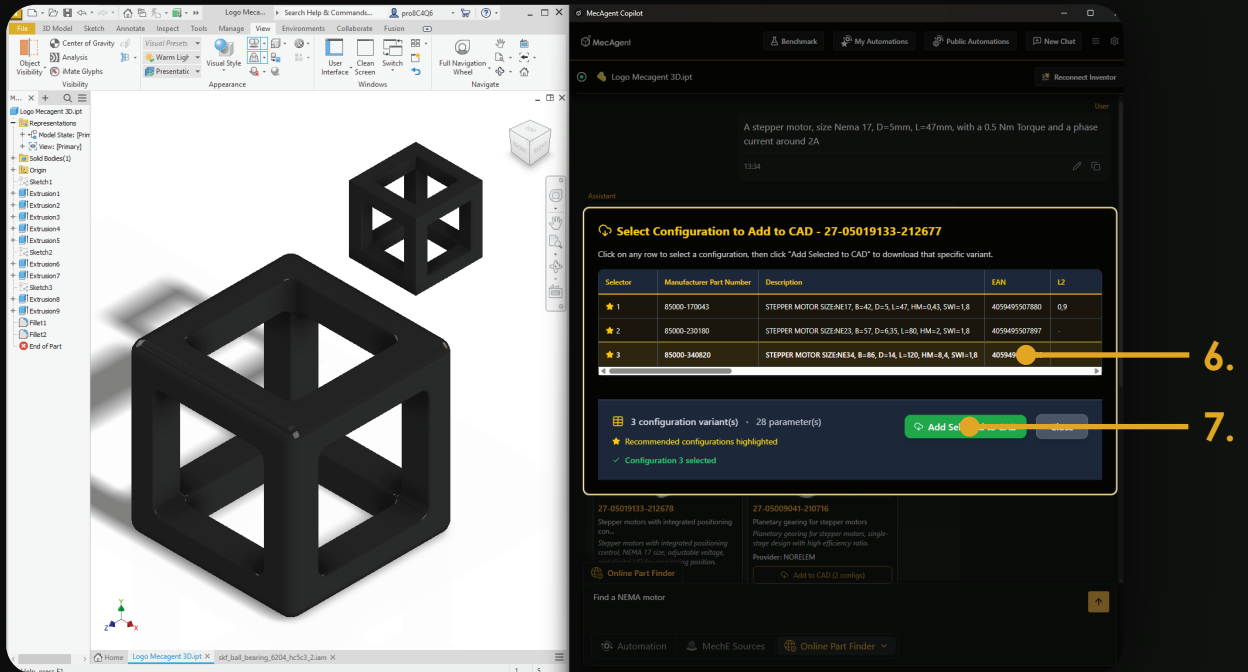
Step by Step



Standard Part AI Finder

How it works ?

Step by Step



5. The AI processes your request to find the part in our product catalog. This takes between 20 seconds to 5 minutes. Once ready, you'll hear a sound notification
6. Select the part that matches your needs.
7. Click "Add to CAD" to see the different configs of the part and choose the one you want and click on "Add Selected to CAD" button.

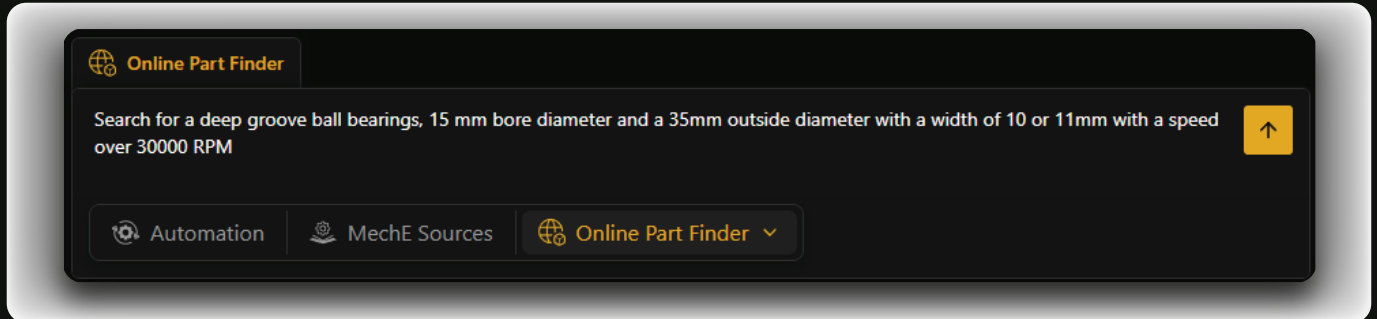
Today, it is possible either to import the part by first saving it to your files and then importing it into your CAD software like any other standard component, or to go directly to the source and download the part from the internet.

This feature is currently in an experimental phase, as not all catalogs are yet categorized within our product database.

Standard Part AI Finder

How it works ?

How to prompt



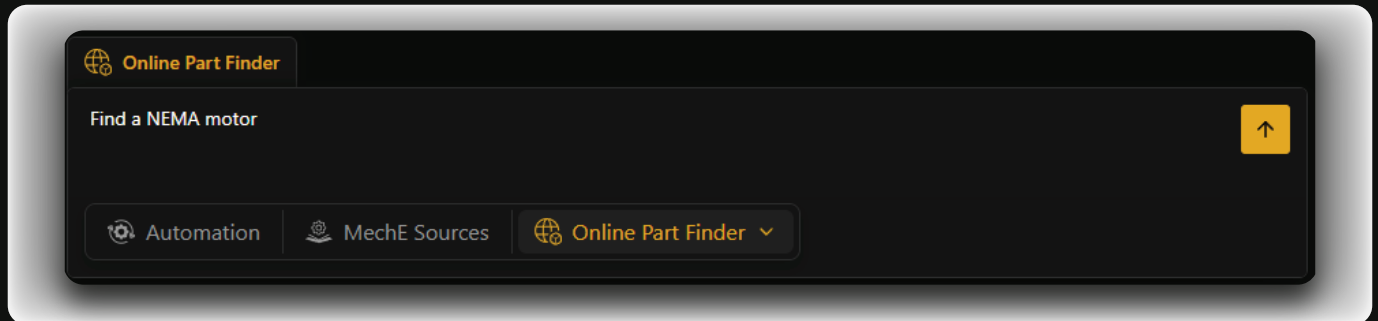
What you can ask

- ✓ "A stepper motor, size NEMA 17, shaft diameter 5 mm, length 47 mm, with 0.5 Nm torque and ~2A phase current"
--
- ✓ "Find a hex bolt M8, 30 mm length, stainless steel, ISO standard"
--
- ✓ "Ball bearing with 20 mm inner diameter and 47 mm outer diameter"
--
- ✓ "Linear rail guide, 15 mm width, with matching carriage"
--
- ✓ "Timing belt pulley for GT2 belt, 20 teeth, 5 mm bore"
--
- ✓ "Identify a component based on function (e.g. "compact gearbox for low-speed high-torque application")"

Standard Part AI Finder

How it works ?

How to prompt

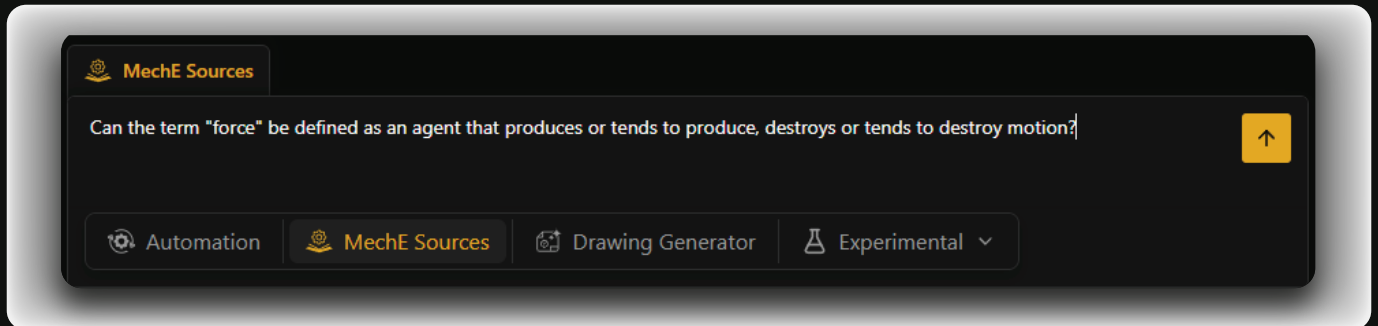


What you should not ask :

- ✗ "Find a NEMA motor"
--
- ✗ "Search with vague or incomplete descriptions without key specifications"
--
- ✗ "Expect guaranteed availability or exact matches across all suppliers"
--
- ✗ "Retrieve highly specialized or proprietary components not publicly indexed"
--
- ✗ "Assume full compatibility validation within your assembly or system"

Standard Part AI Finder

Advanced Prompting Techniques



- Provide the necessary details, such as **dimensions** and **measurements**, to ensure an accurate part.

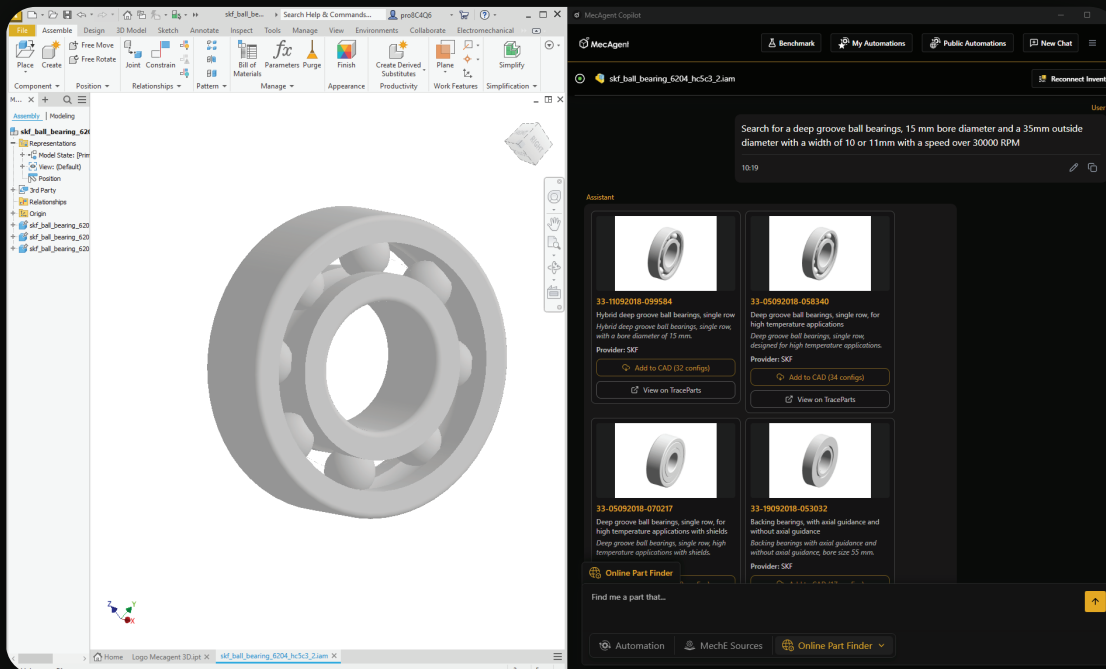
Standard Part AI Finder

Common Examples

Find a ball bearing

Prompt: "Search for a deep groove ball bearings, 14 mm bore diameter and a 47mm outside diameter with a width of 10 or 11 mm with a speed over 30000 RPM"

Waiting time : ~30 s



Standard Part AI Finder

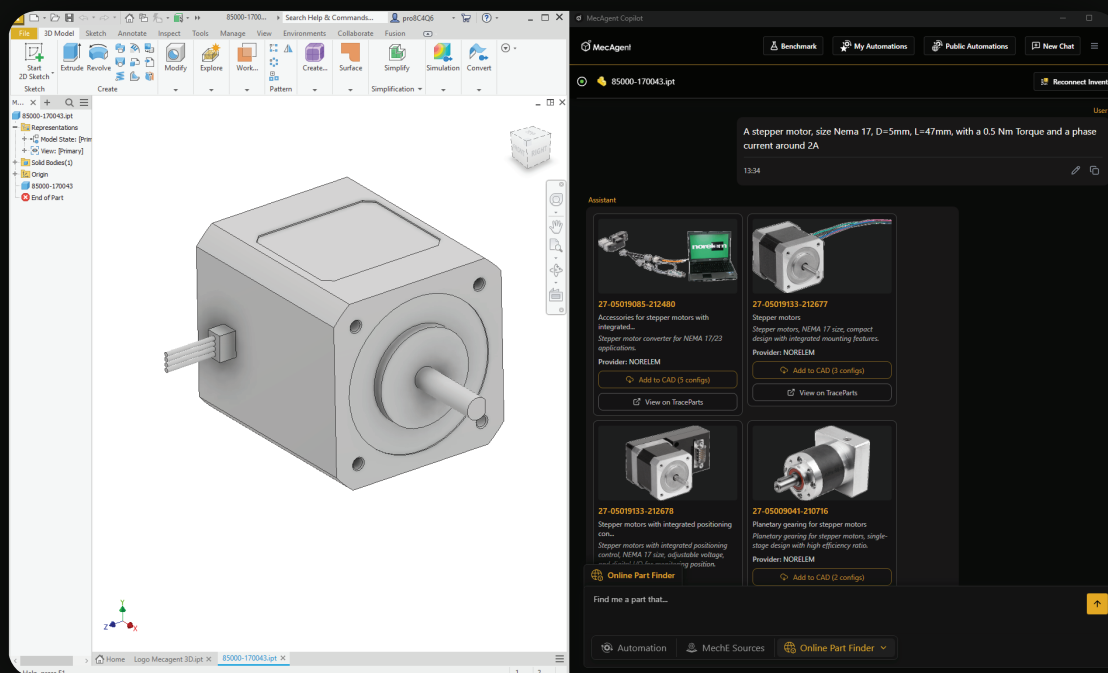
Common Examples

Find a Nema Motor

(<https://youtu.be/L6nWIAVQ8dl?si=r-MPXNp0T9vZ6O2Q>)

Prompt: "A stepper motor, size Nema 17, D=5mm, L=47mm, with a 0.5 Nm Torque and a phase current around 2A"

Waiting time : ~20 s

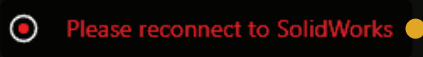


Connection Issues

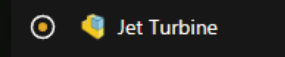
Connection issues

With the CAD Software

From this



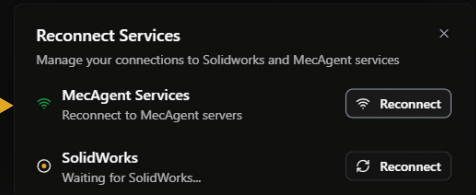
To this



1. Try the Reconnect button:

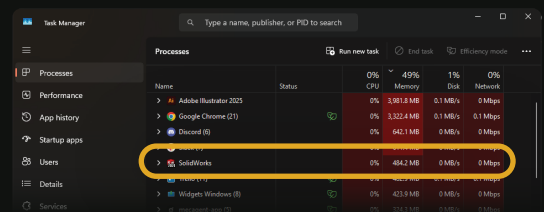


- Click on both Reconnect button



2. If after the last step it doesn't work:

- Open the Windows Task Manager
- Filter "solidworks"
- Kill ALL processes related to SolidWorks
- Launch SolidWorks and allow it to fully initialize
- Click the reconnect button again



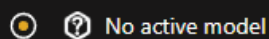
3. If after the last step it doesn't work:

- Run the app as administrator
- Ask for your IT company to authorize the app in the firewall system
- Firewall Configuration Problem:
 - MecAgent requires access to our servers. You may need to configure your firewall to allow the following connection. Contact your IT department or network administrator for assistance with firewall configuration:
 - Domain: proxy.mecagent.com
 - IP Address: 13.219.4.52
 - Port: 443
 - Protocol: TCP

4. If after the last step it doesn't work, book a call :

<https://calendar.app.google/32mqtzNMV5auL2MX6>

Bonus. If you have this



,save your file

Safety & Data

Safety & Data

Safety & Data

We do not collect or store any data related to your assemblies, parts, or files.

The only data sent to our servers are the chat requests used to generate automations and the resulting automation code, which does not include sensitive design information.

All processing is securely handled on AWS infrastructure.

SOC 2 certification is available with the Enterprise plan.

For large enterprises, we can also provide an on-premise solution implemented directly on your own servers. If this is of interest, please schedule a meeting with our team :

<https://calendar.app.google/32mqtzNMV5auL2MX6>

FAQ

Is this available on all CAD software?

For now, it's only available on SolidWorks and Inventor, and it should work with all recent versions. We're planning to make it available on other platforms as well, but we don't have a release date yet.

If you are interested in a demo or want MecAgent to be compatible with another CAD software, please book a meeting with our team :

<https://tally.so/r/VL1yGg>



MecAgent

AI CAD Copilot

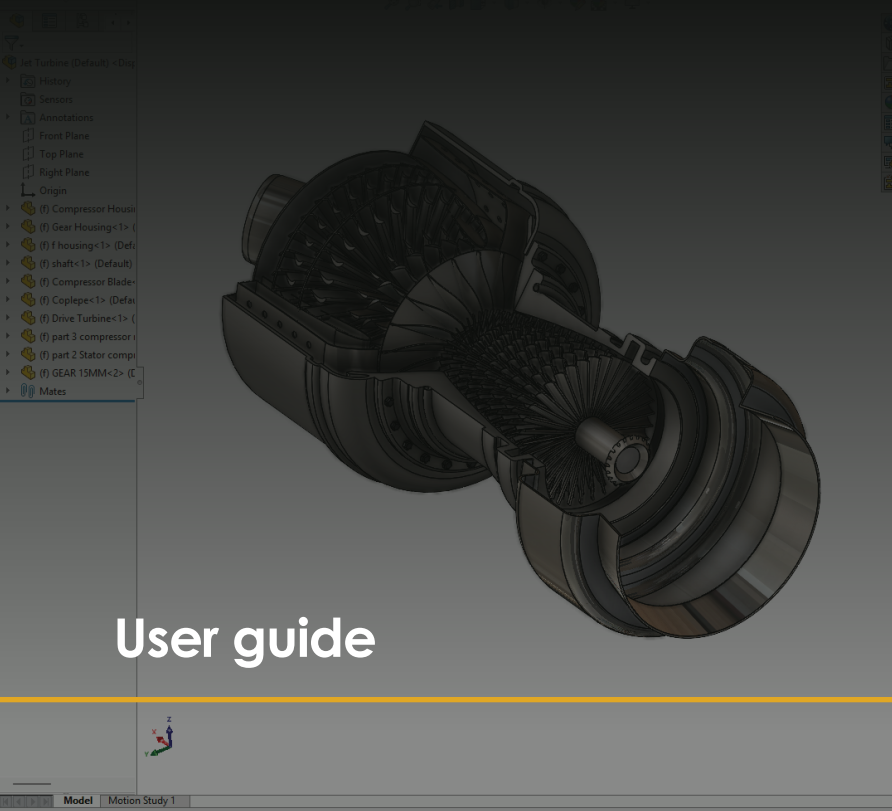
www.mecagent.com

contact@mecagent.com

+33 (0)7 51 64 31 42

2 Marina Blvd B300

San Francisco, CA 94123



User guide

Assistant

Hi! I'm MecAgent. You can ask me to create any automation you need, for example batch exports, auto-fillets, or mass renaming of assemblies. Let me free up your time and save you hours for real engineering.

Apply Fillet on X Axis Bulk Export DXF Multi-Body Part Coloring

Create an automation that...

Automations

Experimental