shapr3D | MANUAL

26.170 v1

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Getting started with Shapr3D

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The basics

Installation, updating, and requirements
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Download Shapr3D

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Shapr3D Basic is free of charge, while Shapr3D Pro requires a monthly or yearly subscription. Before downloading and installing Shapr3D, check our Compatible devices page to ensure your device meets the requirements.

> **Compatible operating systems**
>
> Shapr3D is currently available on Windows, macOS, iPadOS, and visionOS. Android is not supported at this time. For a touch-based experience, explore Shapr3D on a Windows device that supports pen and touch input. Fore more information, visit our Compatible devices page.

Download links

Platform	Installer
iPadOS	App Store installer
macOS	App Store installer
macOS	Direct installer
Windows	Direct installer
Windows	Microsoft Store installer

How to download and install Shapr3D

For detailed instructions on how to download Shapr3D, follow the guide below.

Windows

From the installer

Access the download link in the following ways:

- **Install directly (Recommended)**: Download the .appinstaller file to your computer and launch directly via App Installer file association.
- **Install via the command line**: Download the .appinstaller file to your computer and then use the following command in a PowerShell console:
  `Add-AppxPackage -AppInstallerFile "PATH\TO\Shapr3D.Package.appinstaller"`

> **Note**
>
> These options require that App Installer is installed on your device. Windows versions that have Microsoft Store enabled ship with App Installer available, however if you opted-out of Microsoft Store, please make sure you have the latest version installed.

- Go to shapr3d.com > **Get started for free** > click **Download for Windows** under **Windows.**

From the Microsoft Store

Download Shapr3D from the Microsoft Store via your browser using this link: apps.microsoft.com/store/detail/shapr3d/9N4K9QFV4XFC

You can also follow these steps to download Shapr3D from the **Microsoft Store**:

1. Open **Microsoft Store**.
2. Sign in to the Microsoft Store with your Microsoft account.
3. Search for Shapr3D.
4. Select **Install**.

iPadOS

Download Shapr3D from the App Store: apps.apple.com/app/shapr3d-cad-modeling/id1091675654.

macOS

From the installer

1. Access the download link in any of the following ways:
   - Download the installer here: https://download.shapr3d.com/mac/latest
   - Go to shapr3d.com > **Get started for free** > click **Download for macOS** under **macOS**.
2. Choose a folder to save the **.dmg** file, then click **Save**.
3. Open the downloaded file.

4. Follow the prompt to click and drag **Shapr3D** to copy it into your **Applications** folder.

From the App Store

Download Shapr3D from the App Store via your browser using this link: apps.apple.com/app/shapr3d/id1091675654.

You can also follow these steps to download Shapr3D from the **App Store**:

1. Open **App Store**.
2. Sign in to the App Store with your Apple ID.
3. Search for Shapr3D.
4. Select **Get**.

Read more

If you encounter any issues during installation, visit Installation issues.

Update Shapr3D

Ensure that you have all the latest features available by keeping your Shapr3D software version up-to-date. Follow the guide below to learn how to update Shapr3D based on your device or installation type.

> **Tip**
>
> To learn about the latest version and updates, visit our Shapr3D release updates page or the Shapr3D Changelog.

Windows

From the installer

To update Shapr3D using the installer downloaded from the Shapr3D website, follow the steps for downloading Shapr3D from the installer. When prompted, confirm that you want to replace the older version with the new version that you downloaded.

From Microsoft Store

To update Shapr3D, follow these steps:

1. Open **Microsoft Store**. Make sure that you're signed in to your Microsoft account.
2. At the bottom left, click **Downloads**.
3. At the upper-right corner of the **Downloads** page, click **Check for updates**.
4. If an update is available, Shapr3D automatically updates in the **Updates & downloads** section.
   Note: If you have an older version but don't see Shapr3D in the available updates, follow the steps to update Shapr3D from the website.

iPadOS

From the App Store

1. Open **App Store**. Make sure you're signed in to your Apple ID.
2. Search for Shapr3D.
3. Select **Shapr3D CAD modeling**.
4. Under **Shapr3D CAD modeling**, look for the **Open** or **Update** button.
   - **Open** – This button means that Shapr3D has the latest version available on your device. Tap this button to open Shapr3D.
   - **Update** – This button means there is a new update available. Tap this button to update Shapr3D.

macOS

From the installer

To update Shapr3D from the website, follow the steps for downloading Shapr3D from the installer. When prompted, confirm that you want to replace the older version with the new version that you're downloading.

From the App Store

To update Shapr3D from the App Store, follow these steps:

1. Open **App Store**. Make sure you're signed in to your Apple ID.
2. Search for Shapr3D.
3. Select **Shapr3D CAD modeling**.
4. Under **Shapr3D CAD modeling**, look for the **Open** or **Update** button.
    - **Open** – This button means that Shapr3D has the latest version available on your device. Click this button to open Shapr3D.
    - **Update** – This button means there is a new update available. Click this button to update Shapr3D.

Shapr3D requirements and specifications

Below are the requirements and specifications for Shapr3D on all devices.

Requirements

Compatible devices

- Visit our Compatible devices page to view all recommended system requirements for all devices.
- To learn more about different screen options for touch and pen input, visit these pages:
    - Apple: Compare iPadOS models
    - Microsoft: Compare Surface computers

Shapr3D account

- Before you start using Shapr3D, sign up for an account using a valid email address.

Device specifications

- **Device limit:** 3 devices at a time across all platforms.
  Note: When you log in to 4 or more devices, you will be automatically logged out of the device(s) with the oldest login date.
- **Languages**:
    - English
    - Chinese (Simplified and Traditional)
    - French
    - German
    - Italian
    - Japanese
    - Korean
    - Portuguese (Brazil)
    - Russian
    - Spanish
- **Internet connection:**
    - Modeling: No
    - Download Shapr3D: Yes
    - Update Shapr3D: Yes
    - Use Sync: Yes
    - Access web links in the software: Yes
    - Import from external cloud services: Yes
    - Export to external cloud services: Yes
    - Updating account email preferences: Yes

Subscription

Plans, free licenses, cancellations, and refunds
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Shapr3D subscription plans

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Shapr3D is a freemium app, which means all users have access to every modeling tool and basic feature, forums, and help materials for free. On top of that, all users can work with Shapr3D on iPadOS, macOS, and Windows.

Shapr3D offers the following subscription plans:

- Shapr3D Basic: This is the free version.
- Shapr3D Pro: A subscription-based version designed for professionals, offering advanced features including:
    - Unlimited projects
    - Export formats for all major CAD platforms, high-quality manufacturing, and AR
    - Real-time product renders in Visualization
    - Technical drawings
    - Priority technical support
- Shapr3D Enterprise: A subscription plan tailored for businesses, offering all Pro features plus:
    - Enterprise-compliant security and data management
    - Easy access with team login or single sign-on (SSO)
    - CAD ecosystem importers and exporters (like CATIA, NX, and more)
    - PLM integration
    - Consolidated billing
    - Customized training
    - Dedicated account manager
- Shapr3D Educational License: A free subscription granting access to Pro features for students and educators.

We offer subscriptions instead of perpetual licenses because that's the most beneficial to you:

- **You're in charge:** every month or year, you can choose to renew and keep using Shapr3D.
- **Low cost of entry**
- **Flexible** subscription-handling
- **The latest updates** are always at your fingertips
-

> **Tip**
>
> Visit our **Pricing page** to learn about the cost for each subscription, payment plan options, and included offers.

Shapr3D Basic

Shapr3D Basic is a free version with a lot of accessible modeling essentials, making it an excellent tool for creating models or practicing your CAD skills.

Shapr3D Basic features

With Shapr3D Basic, you have access to the following:

- **Fundamentals:**
  - Access your account across Windows, iPadOS, and macOS platforms
  - 2D sketching and constraints
  - 3D solid modeling
  - Augmented Reality viewer
  - Projects
    Note: You can create a maximum of 2 projects.
- **Shapr3D Cloud:** Shapr3D Sync
- **Import:** All supported files. To learn more, visit Import.
- **Export:** Low-resolution STL and 3MF only. To learn more, visit Export.

Upgrading your plan

You can upgrade your Shapr3D Basic plan to Shapr3D Pro anytime to access more advanced features such as Visualization and 2D Drawings.

Visit Shapr3D Pro to learn how to subscribe, and our Pricing page to learn more about the difference between Shapr3D Basic and Shapr3D Pro.

> **Tip**
>
> If you choose to cancel your subscription or downgrade to Shapr3D Basic, you will still have full access to all features until the subscription expires.

Shapr3D Pro

Upgrade your account to Shapr3D Pro to experience full functionality. With Shapr3D Pro, you can create unlimited projects and gain access to more advanced features such as Visualization and 2D drawings (Technical drawings).

Visit our Pricing page to learn more about the difference between Shapr3D Basic and Shapr3D Pro.

Website vs App Store subscription

You can subscribe from the **Shapr3D website** or the **App Store**. Subscribing from either method will give you the advanced features that Shapr3D Pro offers, but we recommend subscribing via the website for these added benefits:

- You have access to the teams feature.
- You can pay with a company card.
- You can deduct VAT by providing your company VAT number.
- Your full subscription payment goes to Shapr3D without a commission paid out to Apple.

Teams feature

When you subscribe to Shapr3D Pro from the website, you automatically get access to Pro features, including teams.

With teams, you can enjoy the following benefits:

- **Your own team**
  - By default, your account becomes a team that you own.
  - You can choose to use your account individually or purchase additional seats to invite team members to collaborate.
- **Invitations to other teams**
  - Easily join other teams with a simple email invite from team owners.
- **Collaboration**
  - With teams, you and your teammates can share projects in a single space.

Subscribe to Shapr3D Pro

Via the Shapr3D website

1. If you haven't yet, create a Shapr3D account.
2. You can access the subscription page in the following ways:
    - Go to app.shapr3d.com, and log in to your account. Under **Subscription** on your account page, select **Upgrade**.
      Note: You can also log in by going to shapr3d.com > **Sign up**.
    - Go to shapr3d.com/pricing.
3. If prompted, choose **Pro**.
4. Choose between a **Yearly** or **Monthly** payment plan, then select **Subscribe now** or **Start 14-day free trial**.
5. Under **Number of seats**, select how many seats you want to purchase. Go to Seats to learn more.
    - A team member automatically occupies one seat.
    - If you don't plan to invite team members, you can select just 1 seat for yourself.
    - You can still purchase additional seats after subscribing.
6. Under **Credit card details,** enter your card payment details
7. If required, enter your billing details under **Billing information**.
   Note: Billing information may be optional depending on your country if you don't enter a tax ID. If you enter a tax ID, billing information is mandatory.
8. **Optional**: If you have a valid coupon code, click **Add coupon code**. Go to Coupons to learn more.
9. Select **Subscribe now**.
10. Once your payment is successful, you will receive an invoice via email.
    Note: If you encounter issues subscribing from the website, contact support.
11. To check if your subscription is successful, go back to the **Subscription** page in your account to confirm that your subscription has been updated.
    Note: By default, your subscription automatically renews. You can turn off auto-renewal by cancelling your subscription.

Seats

Team owners can add seats and invite members during checkout or purchase more seats and invite more members from the account page after subscribing. A team member automatically occupies one seat.

To learn how to re-assign purchased seats to new members, go to the Manage your account.

Auto-renewal

Only team owners can turn on or turn off auto-renewal for the Shapr3D Pro subscription.

If auto-renewal is turned off before the end of the billing cycle:

- All team members are automatically removed and lose access to the team and their projects at the end of the billing cycle.
- The team owner's plan automatically downgraded to Shapr3D Basic

To learn how to turn on or turn off auto-renewal, go to Manage your account.

Trial

A team trial is free regardless of the total number of seats.

During an active trial:

- Team owners can invite members to the team during the trial period.
- The newly invited team members will be enrolled in the team trial, which will last until the end of the team owner's trial period.

After a trial ends:

- The team owner gets charged for the total amount of seats.

Coupons and discounts

For website subscriptions, Shapr3D occasionally offers discounts and promotions. If you have a valid coupon code, you can add it at checkout.

Coupons you apply at checkout automatically apply to all seats you purchase during and after the transaction. Once the coupon expires, the seats renew at full price.

To add a coupon code:

1. From the payment details screen, click **Add coupon code**.
2. In the **Enter code** field, enter your coupon code, then select **Apply** to continue.
3. To finish subscribing, click **Subscribe now.**

Via the App Store

> **Important**
>
> Unlike a Shapr3D Website subscription, the teams feature is not available through an App Store subscription.

1. If you haven't yet, create a Shapr3D account.
   Note: A Shapr3D account is not the same as an Apple ID.
2. Open **Shapr3D**.
3. If you use Shapr3D Basic, a prompt to start a free trial automatically appears whenever you open Shapr3D.
   Note: If you want to subscribe at a later time, select **Skip & Use Limited Version** at the top.
4. Select **Start Free Trial**. Or, select **See Subscription Details** first to view more details and choose between a **Yearly** or **Monthly** payment plan. To subscribe, select **Start 14-day Free Trial**.
5. Confirm your App Store payment.
   Note: Visit Apple Support to learn more about managing your App Store subscriptions.
6. Once you see a prompt telling you that your purchase was successful, select **OK**.
7. To check if your subscription is successful, log in to your account at app.shapr3d.com and go to **Subscription** to confirm that your subscription has been updated.
8. **Optional**: If you decide to subscribe directly from the website, you can do so under **Subscription** by selecting **Manage** under **Expires on**, then selecting **Change to a direct Shapr3D plan**.

Refunds

How you request a refund depends on where you subscribed to Shapr3D:

Website subscriptions

- If you subscribed to Shapr3D Pro through our website and you're an individual user or Team Owner, you may be eligible for a refund.
- Refunds are possible within 14 days of your subscription date.
- To request a refund, submit a support ticket.

Enterprise licenses

- For Enterprise subscriptions, please contact your account manager for any refund or billing questions.

App Store subscriptions

- If you subscribed through the Apple App Store, only Apple can process refunds for subscriptions made on their platform.
- Shapr3D does not have access to Apple's billing system and cannot issue refunds directly.
- To request a refund, visit Apple Support.

> **Tip**
>
> If you choose to cancel your subscription or downgrade to Shapr3D Basic, you will still have full access to all features until the subscription expires.

Shapr3D Enterprise

Shapr3D offers subscriptions for enterprises.

Excluding account management, the Shapr3D Enterprise subscription works the same way as Shapr3D Pro.

With Shapr3D Enterprise:

- You can engage your team and stakeholders with immersive prototyping for more dynamic product design reviews on the Apple Vision Pro. To learn more, visit Shapr3D on Vision Pro.
- Billing will be custom and managed by the Shapr3D Team.
- All the Enterprise team members can view who is part of their team.
- An account manager will need to approve requests for adding more seats, inviting/removing team members, or modifying the Enterprise subscription.

Shapr3D offers subscriptions for enterprises as well. If you're looking for an Enterprise license, request a quote to kick off the discussion about the pricing.

Shapr3D on Vision Pro

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Overview

Apple Vision Pro is now part of Shapr3D's multiplatform support, joining iPad, Mac, and Windows. Running Shap3D on visionOS allows you to model natively without any supporting devices. Shapr3D on Vision Pro offers the same robust features as other platforms, with the added benefit of an immersive and intuitive spatial CAD experience in its augmented reality (AR) environment.

Note: Apple Vision Pro is available for enterprise users. Learn more.

Immersive experience

Vision Pro's cutting-edge augmented reality brings 3D modeling to life, offering a seamless, immersive experience where you can manipulate and visualize models in a three-dimensional space around you.

Native integration

Shapr3D runs natively on Vision Pro, allowing you to access the full range of features directly within the Vision Pro environment without needing a supporting iPad, Mac, or Windows device.

Full feature availability

Vision Pro offers full access to all modeling features available on other platforms, providing you with the same comprehensive toolkit for designing, editing, and visualizing your projects.

Enhanced team collaboration

With Vision Pro, multiple users can simultaneously view the same model in high-quality AR, allowing you to engage your team and stakeholders with immersive prototyping for more dynamic product design reviews. Learn more how Shapr3D immersive CAD can help your team.

Get started with Shapr3D on Vision Pro

Quickly get started with Vision Pro and Shapr3D with these helpful support pages from Apple:

- Get started with Apple Vision Pro:
  - Adjust the fit of your Apple Vision Pro
  - Turn on and set up Apple Vision Pro
  - Learn basic gestures and controls on Apple Vision Pro
- Download Shapr3D on Vision Pro
- Open Shapr3D on Apple Vision Pro

Shapr3D Educational license

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A Shapr3D Educational license is a free version of Shapr3D — available in select countries — that provides you access to all Shapr3D Pro features. If you're a student, teacher, or faculty member, you can apply for a **free, 1-year educational (EDU) license**. The Educational license is available for renewal as long as you're still part of an accredited educational institution.

Unlike Shapr3D Pro, a Shapr3D Educational license does **not** include:

- Teams feature
- Priority technical support

Choose a topic to learn more:

- Educational license eligibility
- Educational license FAQ
- Apply for an Educational license
- Renew your Educational license

Important notice for Educational applicants

- **Educational use only:** This license is strictly for non-commercial purposes; commercial or professional use is prohibited.
- **Compliance monitoring**: We actively review all applications and usage. Shapr3D reserves the right to cancel any license if abuse or non-educational activity is suspected.
- **Regional availability**: The Shapr3D Educational program is currently available only in select countries.

To learn more, visit shapr3d.com/education.

To learn more about the different subscriptions, visit shapr3d.com/pricing.

Educational license eligibility

To be eligible for the Shapr3D Educational license for one year for free, you must meet the following requirements:

- You must be an active student, teacher, or faculty member of an accredited educational institution.
- You must have an active email address with a school email domain provided by the educational institution.
- You must have a valid school ID with an expiration date issued by the educational institution.
- You have a valid supporting document that proves you are an active member of the educational institution.

Once you complete the requirements above, you can proceed to apply for an Educational license.

> **Tip**
>
> If you've already started a Pro trial, cancel the Pro subscription before you apply for an Educational license.

Educational license availability FAQ

Due to a recent restructuring of the Educational license, you may find that this plan is no longer available in your country. Get some answers below to find out more about this change.

Q: I want to apply for the Educational license but I can see that it's not available in my country anymore. Why is that?

A: Our educational program is currently being restructured and may be temporarily unavailable in some regions. In the meantime, visit our pricing page for other licensing options.

Q: Will the Educational license be available for my country again in the future?

A: We don't have a definite timeline for the relaunch yet, but we'll announce updates if anything changes. In the meantime, visit our pricing page for other available licensing options.

Q: I'm a teacher and I've been teaching Shapr3D for two years now. My Educational license has expired and I noticed that this option is no longer available in my country. I'm in the middle of a school semester and Shapr3D is part of the curriculum. Can you please continue my license as an exception?

A: We understand how important it is to be able to continue with your lessons. As an alternative, you can opt to use Shapr3D Basic which includes all our foundational tools and features for free. For other available options, check out our pricing page.

Q: I'm studying abroad and currently spending my summer in my home country. I'm unable to renew my Educational license from your website. Can you please help?

A: The availability of the Educational license depends on your current location. If you don't see the option while in your home country, we recommend that you renew once you're back in the country of your education.

Q: I have the Shapr3D Educational license and just found out that it's no longer available in some countries. Should I be worried?

A: Your subscription will remain active until it expires. Once your subscription expires and you still have access to renew your Educational license, it means that your country is not affected by the restructuring. You can still continue to use it while you're part of an educational institution. If you have trouble renewing your subscription, you can visit our pricing page for other available options.

Q: Our school uses Shapr3D and now they can't use the Educational license. What alternatives/solutions can you offer?

A: As an alternative, you can opt to use Shapr3D Basic which includes all our foundational tools and features for free. For other available options, check out our pricing page.

Q: I've already submitted the required documents for the Educational license. Are you able to give me a license until I finish university? I only have a year left.

A: Unfortunately, we cannot give you a license if the Educational license is not available in your country. As an alternative, you can opt to use Shapr3D Basic which includes all our foundational tools and features for free. For other available options, check out our pricing page.

Q: My Educational license expired and I can’t renew it. All my projects in Shapr3D are locked and I need them to give presentations. How can I access them again?

A: Shapr3D Basic allows you to access up to 2 projects. You can export your projects to a low-res STL, and then import the projects that you need for your presentation. For other available options, check out our pricing page.

Q: Which countries are affected by this change? Will you restrict more countries in the future?

A: The countries affected by this change may vary depending on the ongoing restructuring. We recommend that you visit our pricing page to check the options available for your country.

Apply for an Educational license

To apply for the Shapr3D Educational license, follow these steps:

1. Make sure you qualify for an Educational license.
2. Create a Shapr3D account using the email address provided by your institution.
3. Verify your account.
4. Go to shapr3d.com/pricing.
5. Select **Free for Students and Educators.**
6. Log in with your Shapr3D account.
7. Fill out all fields on the **Request Educational license** form.

> **Tip**
>
> If you're a faculty member who wants to request to whitelist a batch of email domains, visit Educational license email whitelist guide.

8. For faster processing, make sure to attach the following documents:
    - Student/faculty ID with an expiration date
    - Supporting document that proves you are an active member of the educational institution
9. Select **Send Request**.

Approval timeline

After you apply for an Educational license, the Shapr3D team will review your application. The result will be sent to you via email within 3 to 4 business days.

If your institution is part of the Shapr3D whitelist, your request will be automatically approved within 1 business day.

Approved application

Once your application is approved, you can start using Shapr3D Pro right away.

To get started, visit Shapr3D basics.

Declined application

An Educational application may be declined due to any of the following reasons:

- You applied with a personal email address instead of the email address provided by the educational institution.
- The valid ID you attached has already expired or doesn't have an expiration date.
- You didn't attach any supporting documents to prove that you're an active member of the educational institution. You can attach any of the following documents:
    - Screenshot of your online student portal
    - Signed and stamped document that confirms your active status as a member of the educational institution.

Once you have the complete requirements, you can re-apply at any time.

Renew your Educational license

You can renew your Shapr3D Educational license as long as you’re still an active member of an educational institution.

When your Educational license expires, your account is automatically downgraded to Shapr3D Basic and you'll only have access to the two most recent projects you modified. Once you renew your Educational license or subscribe to Shapr3D Pro, you'll regain access to all your projects immediately.

To renew your Educational license, simply apply for an Educational license again.

Cancellation and refunds

Depending on your method of subscription, you can cancel either from the **Shapr3D website** or the **App Store**.

Cancel via the Shapr3D website

1. Go to app.shapr3d.com and sign in.
   Note: You can also log in by going to shapr3d.com > **Sign up**.
2. Go to **Settings** > **Subscription**.
3. Look for the **Auto-renews on** section, then select **Manage** > **Cancel subscription**.
4. To confirm, select **Cancel subscription**.

> **Important**
>
> By cancelling your subscription you disable automatic renewal when the current subscription expires. Your subscription plan remains active until the end of its term, and your plan details are updated with an expiration date.

5. **Optional**: If you want to turn auto-renewal back on, go back to the **Subscription** page and then select **Turn on auto-renewal** in the **Expires on** section.

Cancel via the App Store

1. Open the **App Store** on your device.
2. Go to your profile > **Subscriptions** > **Shapr3D**.
3. Select **Cancel Free Trial** or **Cancel Subscription**.
   Note: For more information about managing your App Store subscriptions, visit How to switch to a different subscription plan.

Get a refund

If you are a consumer (and not a business user) subscribed via the **Shapr3D website**, you can submit a support ticket within 14 days of purchase and the Shapr3D team will issue a refund for you.

If you subscribed via the **App Store**, contact Apple to check if you are eligible for a refund.

Account and storage

Account management and data security
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Create a Shapr3D account

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Before you can start using Shapr3D, you need to have an account. You can sign up through the website or directly from Shapr3D.

Note: If you installed Shapr3D for the first time, you will be asked to create an account after the onboarding tutorial.

To learn how to create an account, follow these steps:

1. Access the sign up page in the following ways:
    - **From the website**: Go to app.shapr3d.com > **Sign up**.
    - **From Shapr3D:** Select **Get Started**.
2. Fill out the required fields for **Work email**, **Password**, **Re-enter password**, and **Your name**.
3. Toggle the switch to accept the Terms of Use and Privacy Policy, then select **Sign Up**.
4. Verify your account by clicking the verification link that will be sent to your email address. If you don't receive a verification email, check your spam or go to your account page to resend a new link.

> **Tip**
>
> After signing in to Shapr3D for the first time, you'll be prompted to set up your devices and complete the onboarding tutorial. Simply choose the device you'll use for modeling and follow the 6-minute tutorial to get started.

Reset your Shapr3D account password

If you've forgotten your Shapr3D password, you can reset it through the website or directly from Shapr3D. Follow the steps below to learn how.

1. Access the reset page in the following ways:
    - **From the website**: Go to app.shapr3d.com.
    - **From Shapr3D:** Open Shapr3D and then select **Log In**. If you're still logged in, log out first.
2. From the **Log In** page, select **Forgot password?**
3. From the **Reset password** page, enter your email address in the **Email** field.
4. Select **Send Reset Link** to receive an email from the Shapr3D team (no-reply@shapr3d.com).
5. Open the email and select **Reset password** to direct you back to the reset page.
   Note: If you don't receive an email or experience other issues, go to Password reset issues.
6. Enter your new password in the **Password** and **Re-enter password** fields.
7. To finish, select **Save**.

Manage your account, team, and shared projects on the web dashboard

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You can manage your personal and team settings from the Shapr3D web dashboard. This page serves as the central hub for your account, where you can access both your dashboard (for shared projects and published versions) and your account settings (for team and subscription management).

From the web dashboard, you can:

- Access projects shared with you under **Shared with Me**
- View and manage your **Published Versions**
- Update your name, email, or password
- Manage team members, roles, and team settings
- Review your subscription type and billing information
- Switch between teams and see your subscription plan (Basic, Pro, Custom Plan, or Guest)

Access your web dashboard

You can access your account page in the following ways:

- **From the website**: Go to app.shapr3d.com and then log in.
- **From Shapr3D:** Go to **Settings** or **Preferences** > **Manage Subscription**. If prompted, log in to your account.

Log in options

Login with username and password

From the website

1. Go to app.shapr3d.com.
2. Enter your email address.
3. Select **Continue**.
4. Enter your password.
5. Select **Login**.

From Shapr3D

1. Open Shapr3D.
2. Select **Log In**.
3. Enter your email address.
4. Select **Continue**.
5. Enter your password.
6. Select **Login**.

Login with Single Sign-On (SSO)

From the website

1. Go to app.shapr3d.com.
2. Enter your email address.
3. Select **Continue** to access the login flow for your Identity Provider (IdP).
4. Select **Log In**.

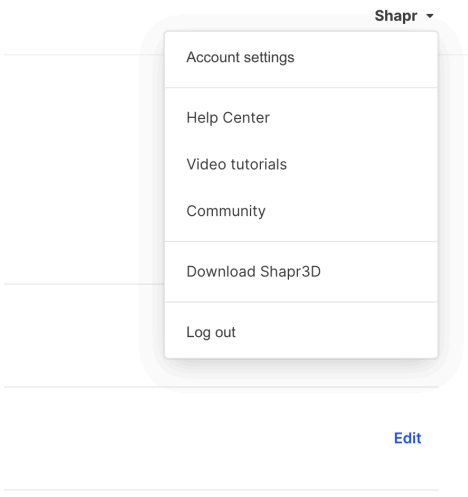
From Shapr3D

1. Open Shapr3D.
2. Select **Log In**.
3. Enter your email address.
4. Select **Continue** to access the login flow for your Identity Provider (IdP).
5. Select **Log In**.

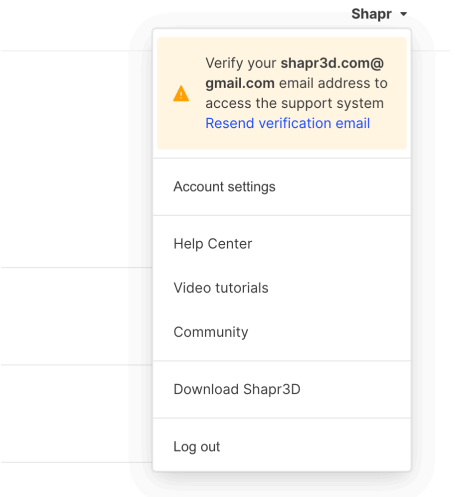
Profile

Once you're logged in, select your profile at the top-right to access the following options:

- **Resend verification email** – This option appears if you still haven't verified your account from the verification email we sent.
- **Help Center** – Takes you to the Shapr3D Help Center.
- **Account settings** – Takes you to the **Accounts settings** page.
- **Video tutorials** – Takes you to the Video tutorials homepage.
- **Community** – Takes you to the **Shapr3D Community** page. Here you can interact with other users and the Shapr3D team. Ask any questions you might have, help other users, or even share your projects.
  Note: You need to sign up for an account to create posts and leave comments in Shapr3D Community.
- **Download Shapr3D** – Takes you to the Shapr3D download page.
- **Log out** – Logs you out of your account.



Verified account



Unverified account

Account settings

Under the account section of the web dashboard, you can manage both personal and team settings.

To access your personal account and team settings, go to your profile > **Account settings**.

Account

From the Account settings page, you can modify the following information:

- **Your Name** – To update your name, select **Update** > Enter the updated name > **Save**.
- **Email** – To update your email address, select **Change** > Enter the new email address > **Send confirmation link**, then log in to your new email address to confirm the new email.
- **Password** – To update your password, select **Change** > Enter your current and new passwords > **Update password**.
- **Two-factor authentication (2FA)** – To add an extra layer of security to your account, you can enable 2FA. Select **Enable** and follow the on-screen instructions to set it up with an authenticator app. After activation, logging in will require both your password and a one-time verification code generated by your chosen authenticator app.
  Note: Make sure to save the recovery code you receive when enabling 2FA, and store it in a safe place in case you lose access to your device or your authenticator app.
- **Delete account** – To delete your account, select **Delete Account**. You can also access this option from Shapr3D by going to **Settings** or **Preferences**.

Members

When you subscribe to Shapr3D Pro from the website, you automatically get access to Pro features, including teams.

With teams, you can enjoy the following benefits:

- **Your own team**
  - By default, your account becomes a team that you own.
  - You can choose to use your account individually or purchase additional seats to invite team members to collaborate.
- **Invitations to other teams**
  - Easily join other teams with a simple email invite from team owners.
- **Collaboration**
  - With teams, you and your teammates can share projects in a single space.

Important

The teams feature is not available for Shapr3D Basic (free version), Educational subscriptions, and App Store-based subscriptions.

Roles in teams

Owner
The owner role belongs to the account that initially signed up. All the data and intellectual property (IP) created within the team belong to the team owner.

Team owners can:

- Name the team
- Manage the team's billing and subscription
- Add and remove seats
- Invite and remove members

> **Important**
>
> Team owners can't be re-assigned and there can't be more than one team owner.

Editor
The team owner can purchase additional seats and invite members to become editors.

Members with an editor role can:

- Access the full feature set of Shapr3D
- Create or import new projects
- Move projects across spaces and folders
- Collaborate in the shared space

Contributor
The team owner can invite up-to 2 contributors to the team free of charge. Contributor access is available when:

- The invited user uses a company email address. Personal email providers (for example, @gmail.com, @outlook.com) and educational domains are not supported.
- There are no Editors in the team. If the team already includes an Editor, contributor roles are not available.

Members with a contributor role can:

- Access the full feature set of Shapr3D with the exception of Drafts
- Create or import new projects to the shared space
- Move projects across folders in the Teampace
- Collaborate in the shared space

Contributors have the following limitations:

- Cannot create new projects
- Cannot export designs
- Only available for users on version **5.831** and later

> **Important**
>
> The contributor role is only available with company domains. Domains like @gmail.com,

@outlook.com or @icloud.com are not supported.

Invite and manage team members as a team owner

Team owners can add or remove members from the team directly.

To add or invite a new member as a team owner:

1. Go to your account page.
2. Optional: If you're a member of other teams, use the dropdown beside the team name to select the name of the team you own.
3. Select **Members**.
4. Select **Invite members**.
5. From the **Invite members** pop-up, enter the email addresses of the members you want to invite.
6. To finish, select **Invite**.

Once a team owner adds a new member:

- The team member gets contributor access.
- A team member occupies one available contributor seat.
- If all contributor seats are filled, the team owner can convert a contributor to an editor to free up a seat.
- The team automatically gets charged for a new seat if all purchased seats are already assigned to a member

If members are removed from the team:

- The removed member's seat will remain available
- The empty seat is assignable until the end of the billing cycle

Remove team members and reassign seats

When you remove a member from your team, they'll lose access to all projects in your team's Spaces and their Drafts. They'll still retain access to any projects that have been individually shared with them. Before removing a member, advise them to back up or transfer any important designs to prevent data loss.

To remove a member:

1. Select **Members** on your account page.
2. Select … beside the team member you want to remove.
3. Select **Remove member**.
4. To finish, select **Remove member** from the prompt.
5. To reassign the empty seats to a new member, simply Invite members again.

Joining a team

If you've been invited to a team, you will receive an invitation email. Simply accept the invitation to join the team and start using Shapr3D right away.

Note: If you can't see the team for which you have accepted an invitation in the Shapr3D Dashboard, make sure to update Shapr3D to the latest version (the teams feature was first released in version 5.650). If the issue persists, contact our support team.

Important

Your team subscription is handled by the team owner who sent you the invitation. If you are removed from a team, any projects stored in your Drafts will no longer be accessible. Ensure that any critical designs are backed up or transferred before leaving the team to prevent data loss. For any issues or assistance with project access, contact Shapr3D Support.

Data ownership

All the data and Intellectual Property (IP) created within a team belong to the owner. When a team member is removed, any projects stored in their Drafts will no longer be accessible to them. To prevent data loss, ensure that important designs are transferred or backed up before removing a member from the team. For any issues or assistance with project access, contact Shapr3D Support.

Subscription

You can view or update your subscription from the **Subscription** page in your account settings.

View subscription information

You can find your subscription type under **Subscription** and **Purchased via**.

Update billing period

To update your billing period from monthly to yearly:

1. Select **Switch to yearly** beside **Billing period**.
2. Check the summary and then select **Switch to yearly** to confirm.

Purchase more seats

To add or purchase more seats:

1. Select **Manage** beside **Seats**.
2. Add the number of seats you want to purchase and then select **Buy seats**.
3. If prompted, enter the details for your card payment and then select **Update & buy seat**.

Manage auto-renewal

When a team owner cancels a subscription, all the team members will be automatically removed and lose access to the team and their projects at the end of the billing cycle.

To turn on or turn off auto-renewal for your subscription:

1. Select **Manage** beside **Auto-renews on**.
2. Select **Cancel subscription**.
3. In the pop-up box that appears, select **Cancel subscription**.

After the cancellation, the subscription will remain active until the end of the billing cycle.

Billing

Team owners can manage the subscription billing information on the Shapr3D account page.

To manage your billing information:

1. Select **Billing** on your account page.
2. Click **Edit** beside the setting you want to update and then make your edits.
3. To finish, select **Submit**.

Team settings

You can update team settings such as your team name in **Team settings** on your account page.

To update your team settings:

1. Select **Team settings** on your account page.
2. Click **Edit** beside the setting you want to update and then make your edits**.**
3. To finish, select **Save**.

Shapr3D web dashboard

Projects in Shapr3D can be shared at two levels. This structure provides flexibility for secure collaboration across full team spaces or individual shared projects.

- **Space-level sharing (Spaces)**:
  All members of the same Space can view and edit projects within it. Projects sync automatically across all members' accounts and devices.
- **Project-level sharing (Shared with Me)**:
  Projects shared directly to your email appear under Shared with Me. You'll have access only to the shared project and not to other projects in the team's Space.

Shared with me

The **Shared with Me** section lists projects that others have shared directly with you. This section helps you keep track of all shared work in one place, so you can easily collaborate on designs without mixing them with your personal or team projects.

Note: Projects appear in the **Shared with Me** section if you've been granted edit access through your email. Projects shared with you as view-only don't appear in this list and you can only open those using the shared link you received.

For each project, you can find:

- **Project name** – The name of the shared project
- **Link** – A direct link to open the project
- **Last updated** – When the shared project was last modified
- **Updated by** – Who last made changes to the project
- **More options**: Click the three dots to:
  - Open the project details on the web
  - Open the project details in a new tab
  - Copy the link for quick sharing or reference

Published Versions

You can keep track of all your shared projects in one place under **Published Versions**. This section lists every project version you have published, so you can quickly find, copy, open, or unpublish links without searching through individual projects.

For each project, you can find:

- **Project name**: Shows which project the link belongs to.
- **Link**: Copy the published link to share it quickly.
- **Location**: Shows where the project is saved.
- **Last updated**: Shows when the link was last updated.
- **Updated by**: Shows who last updated the project.
- **More options**: Click the three dots to:
  - Open the project
  - Open it in a new tab
  - Copy the link
  - Unpublish the link if you no longer want to share it

Getting started as a contributor in a Shapr3D team

If you've received an invitation to join a Shapr3D team as a contributor, here's how to get started.

Accept your invitation

You'll receive an invitation email from the team owner. To accept the invite:

1. Open the invitation email.
2. Select **Accept invite**.
3. Sign up with the email address the invite was sent to.
4. If you already have a Shapr3D account, just log in with your credentials.

> **Important**
>
> Contributor access is only available to company email addresses. Invitations sent to personal emails (such as @gmail.com, @icloud.com) or educational addresses (such as .edu) will not work.

Download and install Shapr3D

After signing up or logging in, install Shapr3D on your preferred device. To learn how, visit Download Shapr3D.

Access a team you joined

When you open Shapr3D, the Dashboard opens in your personal team by default. If you've joined another team, use the team switcher to access that team's shared space.

As a contributor or editor, you can view and collaborate on projects in any shared space you're part of. Each team has its own spaces for shared projects and personal drafts, and access depends on your role.

To learn more, visit Shapr3D Dashboard and Manage your personal account and team settings.

What you can and can't do as a contributor

As a contributor, you can:

- Access the full feature set of Shapr3D (excluding Drafts)
- Open and work on projects in the shared space
- Collaborate with other team members

As a contributor, you can't:

- Create new projects
- Export projects or drawings
- Access Drafts

Note: Contributor access is only available for Shapr3D version **5.831** and later. If you don't see the team or encounter access issues, make sure your Shapr3D version is up-to-date.

Read more

To get started with Shapr3D, check out our Introducing Shapr3D basics tutorial series.

Shapr3D Cloud

Cloud and storage
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Introduction to Shapr3D Cloud

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Shapr3D Cloud is a set of optional cloud-based features that are automatically enabled when you sign up for Shapr3D. With Shapr3D Cloud, you can use features like Sync to make your project available on all your devices, and also provide backup by securely uploading your projects to the cloud.

Using Shapr3D Cloud doesn't mean that you can only access your projects while online. By default, all projects that you create on your device still exist locally and are exclusively tied to your account, allowing you to use Shapr3D and access your projects offline.

Note: If you have Shapr3D Basic, you can only export your projects to low-resolution STL and low-resolution 3MF.

Shapr3D Cloud offers the following features:

- Shapr3D Sync and backup
- Share Published Versions

Shapr3D Sync

Shapr3D Sync is a Shapr3D Cloud feature that keeps your projects up to date and accessible across devices. Personal projects sync to your account, while team projects sync across all members of the team. All synced projects are backed up in Shapr3D Cloud and can be downloaded to work offline. Shapr3D Cloud ensures that your data is yours, giving you a secured solution following industry best practices.

Shapr3D Sync is the solution you need if:

- **You want to access and edit your projects across multiple devices**. With Shapr3D Sync, your projects are always up to date, so you can move between your desktop, tablet, or a newly set-up device and continue where you left off.
- **You want to collaborate in a shared team space**. Team projects automatically sync across all team members, making it easy to stay aligned and work together in real time.
- **You want to keep a cloud backup of your projects**. All synced projects are stored in Shapr3D Cloud, so your data stays safe even if something happens to your device.

Using Shapr3D Sync

To work with projects using Shapr3D Sync, keep the following in mind:

- You need an active internet connection to sync or download project data.
- Projects are stored in the cloud. You must be online to access any project that hasn't been downloaded to your device.
- Offline access is only available for projects that have already been downloaded.

Manage project storage

Shapr3D automatically syncs your projects to Shapr3D Cloud so you can access the latest version from any device you're signed in to.

Each project can be in one of three states:

- **Local only** – Stored only on your current device, not yet backed up to the cloud.
- **Synced** – Stored in the cloud and also downloaded to your device for local access.
- **Cloud only** – Stored in the cloud but not currently downloaded to your device.

You can manage storage individually for each project in the Dashboard.

To manage storage for a project:

1. Right-click (secondary click) the project in your Dashboard.
2. Select **Download Now** or **Remove Download**.
    - **Download Now** – Saves a local copy of the project so you can open and edit it offline.
    - **Remove Download** – Clears the local copy from your device to save storage space. The project stays safely in the cloud and can be downloaded again anytime.

> ### Tip
>
> Projects that aren't stored on your device are still safely saved in the cloud. Just download them again whenever you're ready to work.

Share Published Versions

Published Versions are part of Shapr3D Cloud and allow you to share detailed, navigable views of your projects through a secure web link. Recipients can open the link in any modern browser to explore your 3D designs and 2D Drawings.

2D and 3D compatibility

Published Versions support both 3D models and 2D Drawings, giving stakeholders a complete view of your design. This flexibility makes it suitable for reviews and discussions at any project stage, with the added option to download 2D Drawings as PDFs for offline use.

Interactive collaboration

Team members can add comments directly to specific parts of the design. This keeps feedback focused and visible in one place, simplifying review cycles.

Secure sharing

You control who can view your published version. Set link access settings to limit visibility to your team, specific invitees, or anyone with the link.

AR-ready previews

When opened on supported mobile devices, published versions allow you to preview your model in Augmented Reality (AR). This gives stakeholders a clearer understanding of scale and fit in real-world contexts.

> **Read more**
>
> For details on how to generate and manage a Published Version, visit Published Versions.

Local storage

To ensure smooth cross-platform usage and that the software upgrades safely, Shapr3D stores projects in an internal database. While your projects are stored locally on your device and can be accessed offline, they are not accessible directly over the file system of your device. To use your projects in third-party applications, you need to export them first in one of the supported file formats

Note: If you're using Shapr3D Basic, you can only export your projects to low-resolution STL and low-resolution 3MF.

With Shapr3D Cloud, your projects are automatically backed up via Shapr3D Sync.

Read more

To learn more, visit Shapr3D Cloud and Shapr3D Sync.

Shapr3D basics

Basic navigation, keyboard shortcuts, peripherals, and managing projects
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Modeling concepts

3D modeling concepts overview
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3D CAD modeling concepts

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Overview

3D modeling is the process of creating three-dimensional representations of objects using 3D modeling software. With a 3D CAD modeling tool, you can create detailed designs with depth, visualize concepts with realism, and produce models for real-life simulations.

CAD is a subset of 3D modeling which is a crucial component in manufacturing. If your goal is manufacturable designs, CAD provides a holistic design environment that allows for both 2D and 3D representations. The level of detail you can do with CAD modeling is essential for better collaboration, precise engineering, analysis, documentation, and visualization throughout the product development lifecycle.

Shapr3D is a CAD software solution that offers a complete design workflow from ideation to manufacturable models. Seamlessly transition from concept to detailed 3D modeling and documentation with the help of these main benefits:

- **Integrated workflow.** The multiplatform availability for Windows, macOS, and iPadOS, and Sync capabilities provide you the flexibility to work across various devices, giving you a much more integrated approach to your design process.
- **Offline use**. Shapr3D is fully operational offline across all platforms, enabling you to work on your designs even without an internet connection.
- **Collaboration.** Published Versions are a quick and efficient way to share your concepts, making collaborative reviews effortless.
- **Software compatibility.** Shapr3D works smoothly with other CAD tools. With its support for universal CAD file formats such as STEP and IGES, you can conveniently integrate your designs into other relevant software.

History-Based Parametric Modeling

CAD models are built up through a series of steps. History-Based Parametric Modeling, or Parametric Modeling, is a feature that provides you with the history of your model by recording the series of steps in a timeline or feature tree. As these steps are recorded, connections through dependencies are formed, allowing you to review or modify each step cohesively.

In Shapr3D, you can access the history of your model through the **History** sidebar. As you continue to build your model, the history establishes a connection through dependencies and relationships between different features, dimensions, and constraints. With this existing connection, modifications or iterations you make to

any step automatically propagate through the entire history, making the iterative process much easier and faster.

By keeping the History sidebar closed, you can use Shapr3D just as you would normally without the history feature. However, keep in mind that steps are still recorded in the background so if you want to use history later on, there might be broken connections that you would need to fix.

To learn more about error messages in History, visit Error messages in History.

Direct modeling

Direct modeling, also known as push-and-pull modeling, is a 3D CAD method that focuses on directly manipulating models by pushing and pulling selected topologies of a geometry. With direct modeling, you can adjust geometry quickly without thinking about existing features and constraints. For example, if you want to increase the height or depth of a box, you can simply “pull” any of its faces upwards or sidewards. The same applies to cylindrical bodies or holes: just select the cylindrical faces and then adjust the diameter by pushing and pulling them.

In Shapr3D, you can use a direct modeling approach by simply keeping the History sidebar closed. This approach can provide you the flexibility to make quick modifications without referring to the history timeline.

For more in-depth information on direct modeling, visit How does direct modeling actually work?

Modeling in Shapr3D

When you model in Shapr3D, steps you create are recorded in the History sidebar – essentially providing you with a full-length undo and redo queue for all your actions. The History sidebar keeps a history tree timeline with details of the connections you create within your model. These connections can exist between dimensions, sketches, and bodies of your design-build history. With History, you have the flexibility to quickly review and make modifications to your previous edits.

When you make modifications either directly on your model or from the **History** sidebar, the entire history tree automatically re-adjusts all applicable steps to keep the connection of edits you made in your model. To learn how to use the History sidebar, visit History.

If you don’t need to change how the model was built originally, you can simply keep the History sidebar closed. To learn about the concepts of direct modeling, visit How does direct modeling actually work?

Sketching in Shapr3D

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Sketching is the foundation of 3D modeling in Shapr3D. You can create precise 2D sketches that define the shape and structure of 3D bodies. With constraints, dimensions, and snapping options, you can accurately control your geometry and maintain design intent.

Shapr3D's sketching and modeling workflow follows familiar CAD principles while offering a more intuitive and streamlined experience. You can directly interact with geometry and apply constraints or dimensions as needed. This approach simplifies creating and modifying 3D bodies with tools like Extrude (commonly known as the Push/Pull method), Loft, and Revolve, allowing you to refine designs efficiently while keeping sketches adaptable for future modifications.

Understanding sketch items

Sketches in Shapr3D consist of various 2D items, including lines, arcs, circles, and splines. These items can be freely drawn or precisely defined using numerical input, constraints, and dimensions.

When creating sketches, it's important to distinguish between:

- **Closed sketches** – Fully enclosed shapes that form a **sketch profile**, which can be extruded, revolved, or used as cut profiles.
- **Open sketches** – Unenclosed shapes that can be used for sweeps, guide rails, or projected curves.

Sketch constraints, dimensions, and sketch states

Sketches in Shapr3D can be precisely controlled using **constraints** and **dimensions**. These tools help maintain relationships between sketch items and ensure accuracy in your designs.

Constraints

Constraints define geometric relationships between sketch items. They automatically adjust the sketch when modifications are made, maintaining consistent proportions and alignment.

To learn more, visit Constraints overview.

Dimensions

Dimensions specify exact measurements for sketch items, such as lengths, angles, and distances. Adding dimensions allows you to define precise values, ensuring your sketches remain accurate and scalable.

To learn more, visit Defining and calculating numerical values.

Sketch states

The state of a sketch indicates whether it is fully constrained or still has undefined elements.

In Shapr3D, sketches can be in one of two states:

- **Fully defined:** A fully-defined sketch has all its geometric relationships and dimensions constrained, preventing any sketch items from moving independently. The sketch remains fixed unless a constraint is removed, a dimension is modified, or a parameter is adjusted.

- **Under-defined:** An under-defined sketch has sketch items that are not fully constrained, allowing them to move or change size. You can apply additional constraints and dimensions to control the geometry and fully define the sketch.

> **Tip**
>
> To quickly check a sketch's state, look at the color of the sketch items. Green means fully defined, while blue means under-defined.

Sketching workflow

When sketching in Shapr3D, follow these general steps:

1. Select a sketch plane or face to draw on.
2. Use sketch tools like Line, Circle, or Spline to create the desired geometry.
3. Apply constraints and dimensions to define relationships.

> **Tip**
>
> **You** can create sketches with varying levels of definition, but fully defining them ensures stability and predictable parametric behavior when making adjustments.

Using sketches to create 3D bodies

Once a sketch is complete, you can turn it into a 3D body using Shapr3D's modeling tools. For example, use **Extrude** to add depth to a shape, or use **Revolve** to create cylindrical or spherical shapes, such as bottle bodies, shafts, or rounded enclosures.
Sketches also play a key role in more complex workflows:

- **Loft and Sweep**: Use sketches as profiles and guide curves to create smooth, organic shapes for ergonomic grips or fluid containers.
- **Offset and Projection**: Generate precise outlines by offsetting existing sketches or projecting edges onto new planes.
- **Cutouts and Details**: Create cut features like vent openings, custom engravings, or embossed text directly from sketches.
- **Construction Geometry**: Use sketches to define centerlines, reference curves, or alignment guides for precise feature placement.

To learn more about the available Shapr3D tools for creating 3D bodies, visit the Tools menu.

> **Learn more**
>
> If you're new to Shapr3D, check out our video tutorials to see sketching in action.

Navigation

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Shapr3D Dashboard

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The **Shapr3D Dashboard** is the default screen of Shapr3D.

In the Dashboard, you can:

- Create, organize, and open projects
- View recent Shapr3D updates
- Switch between teams
- Access learning content
- Access Shapr3D settings

Dashboard sidebar

You can find the Dashboard sidebar at the left side of the Dashboard.

Team name and switcher

The team name at the top of the sidebar shows which team dashboard you're currently in. If you're a member of multiple teams, you can switch between them to view different projects.

Opening the team switcher lets you:

- Switch between teams (if you're a member of more than one)
- View each team's subscription type (e.g. Basic, Pro, Enterprise)
- Access Settings or Preferences.
- **Log out** of your Shapr3D account

> **Tip**
>
> To use the full feature of teams, make sure to enable Shapr3D Sync.

To switch between different teams:

1. Select the arrow beside the team name.
2. Select the team name you want to switch to.

Invite members

Under the team name, you can click Invite Members... to start inviting members to your team. To learn more, visit Manage your personal account and team settings.

Search projects

Use the search bar in the Dashboard sidebar to quickly locate a project by name. As you type, the results update in real time, helping you jump straight into the project without manually browsing through Recents or Spaces.

Recents

Under **Recents**, you can find the following:

- **What's New?** – Displays teasers of new features, tutorials, demos, and workflows by industry experts.
- **Recent Projects** – Where you can easily find and open your most recent projects**.**

Shared with Me

The Shared with Me section displays projects where you have edit access that others have shared with you, so you can manage shared work without switching between emails or links.

Published Versions

Click **Published Versions** on the Dashboard to open the Published Versions page in your account. Here you can see a list of all your projects with published links, copy or open a link, or unpublish it when you no longer want to share it.

To learn more, visit Published Versions.

Drafts

Use Drafts to keep early-stage or personal projects separate until you're ready to move them to a Space.

- Projects in Drafts sync only across your own devices and aren't visible in any Space.
- You can create new projects directly in Drafts or move projects between Drafts and your Space.
- If collaborators have been added via their email address, they'll continue to have access to the project even when it's in Drafts.

Spaces

A Space is where you organize and collaborate on projects with your team. While a Space can be shared with others, it can also be used privately if you're the only team member.

In a Space:

- You can create folders, import projects, and build a shared project library.
- The number of members in the Space appears under its name.
- Projects and folders in the Space sync across all members' accounts and devices.
- All members of a shared Space can view and edit projects.
- You can import team projects, archived designs, part libraries, and other content to the Space to share with teammates.
- You can move projects from your Space to Drafts to temporarily work on them privately. Only you (and explicitly added collaborators) will retain access when the project is moved to Drafts.

> ### Tip
>
> Projects should be opened by one person at a time. Opening a project simultaneously from two different devices or accounts is not recommended because it may introduce Sync conflicts.

Learn

From the Dashboard sidebar, you can access the following:

- **Tutorials** – Watch tutorials to hone your Shapr3D skills. These videos range in level of expertise, from beginner to advanced, so there's something for everyone.
- **Workflows** – Learn about the workflows of featured CAD professionals.
- **Stories** – Check out how other Shapr3D users are implementing Shapr3D in their projects and workflows.
- **Manual** – Directly access Shapr3D Manual to quickly search for detailed instructions and information.
- **Forum** – Access Shapr3D Community to interact with other users and the Shapr3D team. Ask any questions you might have, help other users, or even share your projects.
  Note: You need to sign up for an account to create posts and leave comments in Shapr3D Community.
- **Settings (iPadOS and Windows only)** – Manage your account details, subscription plans, and preferences. You can find more details in Settings or Preferences.

Dashboard top bar

You can create, import, and organize your projects from the top bar. To learn more, visit Manage your projects.

> ### Read more
>
> To learn how to manage your personal account or team settings, visit Manage your personal

account and team settings.

Shapr3D modeling space

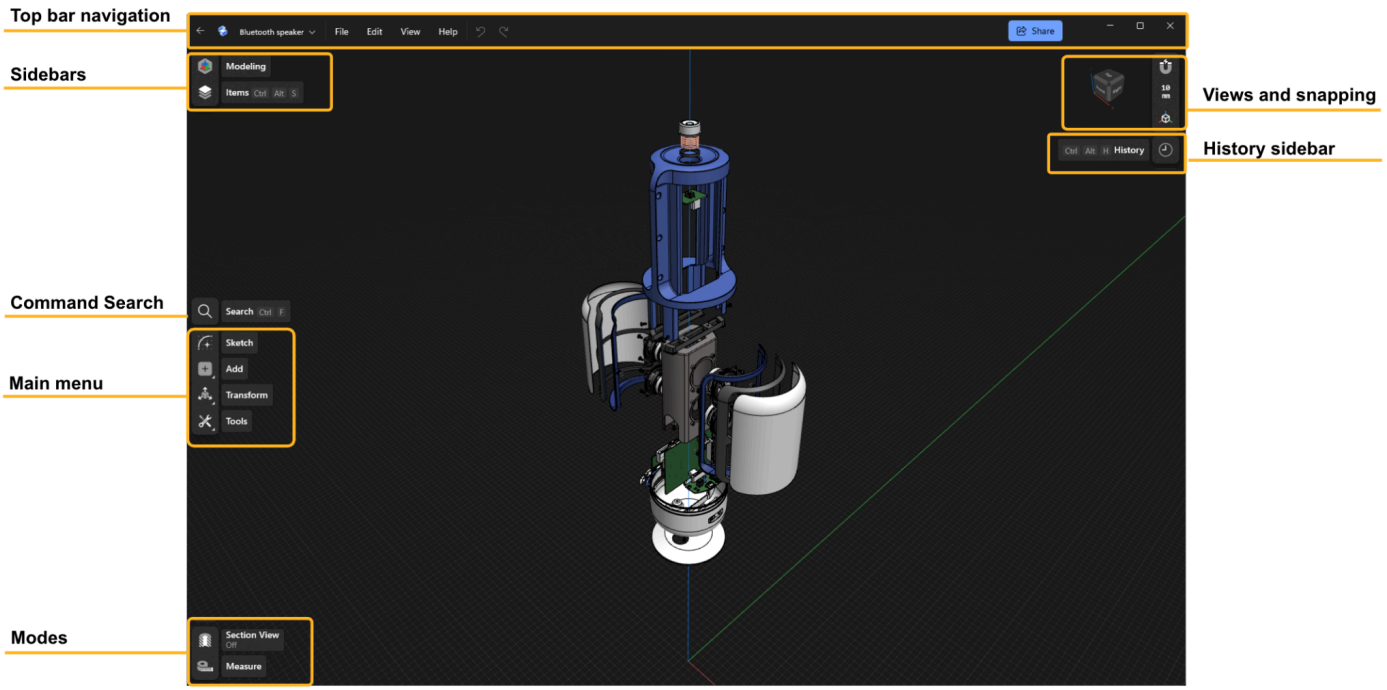
[Article link | Table of contents | Back | Next]

This is your guide to the Shapr3D modeling space user interface.

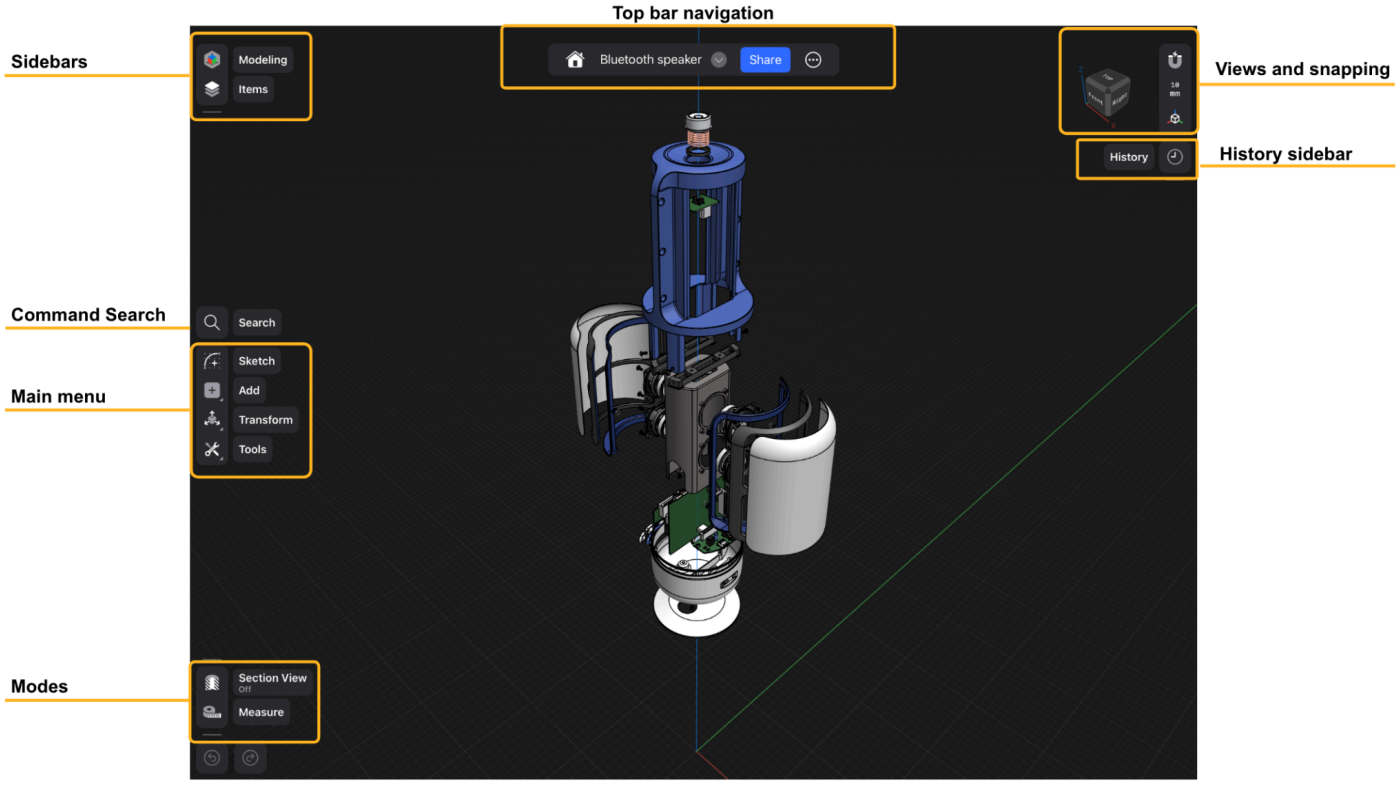
In the modeling space, you can find the following areas:

- Top bar navigation
- Sidebars
- Main menu and Command Search
- Modes
- Views and snapping

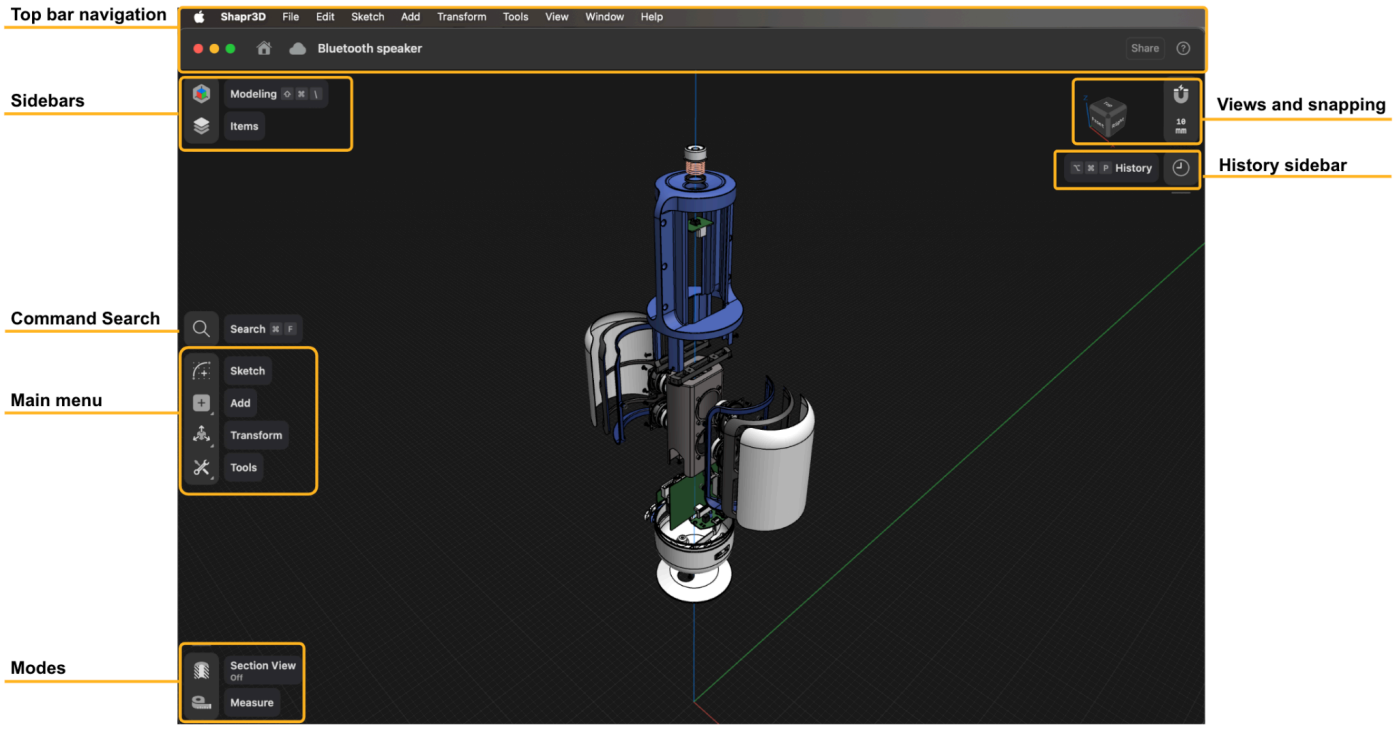
Windows



iPad


Top bar navigation
Sidebars
Modeling
Items
Bluetooth speaker
Share
Views and snapping
History
History sidebar
Command Search
Search
Main menu
Sketch
Add
Transform
Tools
Modes
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Top bar navigation

The top bar navigation is where you can access most settings and basic controls. From left to right, you can find the following:

- **Back to Dashboard (Windows)** ← – Brings you back to the **Dashboard**.
- **Home (iPadOS/macOS)** – Brings you back to the **Dashboard**.
- **Project name and Sync** – Where you can find and rename your project name and access **Sync**.
- **File menu** – Opens the basic file commands such as **New project**, **New Drawing…**, and **Export to…**
- **Edit menu** – Opens the basic edit commands such as **Undo**, **Redo**, **Select all**, **Deselect all**.
- **View menu** – Opens the basic view commands related to your grid position field of view, and more.
- **Help menu** – Opens the basic help settings such as keyboard shortcuts, tutorial videos, and link to Shapr3D Manual and Shapr3D Community.
- **Undo** – Shortcut to the **Undo** command, where you can revert your previous interaction.
- **Redo** – Shortcut to the **Redo** command, where you can redo your previous interaction.
- **Share** – Has a **Share** tab where you can share your project in different ways.
- **Help** – Shortcut to the **Help** menu.
- **More Menu** ••• – Contains the options for **Import**, **Export**, and **Settings.**

Sidebars

The sidebars section is where you can access the Project Sidebar and Items Manager. The Project Sidebar is where you can easily jump between the different spaces and access relevant shortcuts.

- **Modeling** – What you see while in the modeling space. This opens the project sidebar where you can easily navigate between the modeling, Visualization, and 2D Drawings space.
- **Visualization** – What you see while in the Visualization space. This opens the project sidebar where you can easily navigate between the modeling, Visualization, and 2D Drawings space.
- **Drawings** – What you see while in the 2D Drawings space. This opens the project sidebar where you can easily navigate between the modeling, Visualization, and 2D Drawings space.
- **Items** – Shows the **Items Manager**.

Main menu and Command Search

The main menu displays the Command Search and modeling tools. In the default state, only the icons are visible and depending on your preferred setting, you can hover your mouse over the icons to see their names.

From top to bottom, you can find the following:

- **Search** – Opens the command search box.
- **Sketch** – Opens the sketch and constraint tools menu. To learn more about each sketch or constraint tool, visit Sketch menu.
- **Insert** – Opens the Insert menu. To learn more about each tool, visit Insert menu.
- **Construct menu** – Opens the Construct menu. To learn more about each tool, visit Construct menu.
- **Transform** – Opens the transform menu. To learn more about each tool, visit Transform menu.
- **Tools** – Opens the tools menu. To learn more about each tool, visit Tools menu.

Modes

The selection-based mode area is where you can find modes that you can turn on or turn off. The menus in this area adapt to your selection so it only shows you the relevant modes that you need.

- **Section View** – Turns on and turns off Section View.
- **Isolate** – Turns on and turns off Isolate Mode. This mode is visible while using Items Manager and other relevant tools.
- **Measure** – Turns on Measure Mode where you can select items to measure.

Views and snapping

- **Orientation Cube**  – A quick way to navigate the orientation view of your project. To learn more, visit **Orientation Cube**.
- **Snaps / Guides**  – Opens the snapping options for the grid, guide lines, and guide points.
- **Units**  – The place to set your desired unit, lock your grid, and customize specific format settings with inches and degrees. This # icon also displays the current grid resolution of your modeling space, which changes according to the level of zoom.
  Note: For macOS, you can lock your grid from the **View** menu in the **Mac menu bar**.
- **Views/Appearance**  – Opens **Views** and **Appearance**.

Items Manager

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The **Items Manager** lists all items within a project and provides several functions that can keep your project organized and help you make selections.

- Accessing the Items Manager
- Items Manager icons and structure
- Items Manager Functions

Accessing the Items Manager

From the sidebars, select **Items**  to show or hide the **Items Manager**.

Items Manager icons and structure

Every sketch, body, plane, axis, image, and 2D Drawing you create in your project is defined as an item and is listed in the **Items Manager**. You can quickly identify the different types of items in the **Items Manager** panel by the icons displayed next to the item name. See the table below for more information on the icons used in the Items Manager.

Body	
Sketch plane	
Construction Pplane or axis	
MeshConstruction axis	
Image	
Folder	

Items Manager functions

In the **Items Manager**, you can:

- Make selections
- Adjust image opacity
- Create and use folders
- Rename an item
- Hide/show items
- Zoom to an item
- Delete an item

Make selections

The **Items Manager** is another option to make selections in your project. You can define a selection set in the **Items Manager** panel and launch tools from the adaptive menu. To make selections easier, you can filter items in the **Items Manager** by selecting **All Items** at the top and choosing an item type.

> ### Tip
>
> If you're using a keyboard and mouse/trackpad, hold the CTRL or Command key to select more than one item or press the Shift key to select a range. For more information on making selections in Shapr3D, visit Selecting geometry.

Adjust image opacity

You can adjust the opacity of an image from the **Items Manager** by clicking or tapping the percentage value beside the image to open the **Opacity** slider. Use this slider to adjust the transparency of the image.

Create and use folders

You can create folders in the **Items Manager** to organize your projects and group together selection sets.

To create a folder, select  at the bottom of the **Items Manager**.

Once a folder is created, you can add items to the folder using drag and drop.

Rename an item

You can rename an item from the Items Manager by tapping and holding an item with a pen or secondary clicking (right-clicking) with a pointer to an item, then selecting **Rename.**

Hide/show items

Each item in the **Items Manager** has a **Visibility** icon that can be toggled to identify and control the visibility of project elements. Select  to hide an item or  to show an item.

> **Tip**
>
> You can select the Visibility icon of folders to change the visibility of all items within a folder.

Zoom to an item

You can zoom to an item from the **Items Manager** by tapping and holding an item with a pen or secondary clicking (right-clicking) with a pointer to an item, then selecting **Zoom to** (iPadOS or macOS) or **Zoom to Selection** (Windows).

Delete an item

You can rename an item from the **Items Manager** by tapping and holding an item with a pen or secondary clicking (right-clicking) with a pointer to an item, then selecting **Delete**.

Reveal specific items in the sidebar

You can use **Reveal in Items** to quickly locate a part of your model in the Items sidebar.

Right-click (secondary click) a part of your model in the modeling space, then select **Reveal in Items** to highlight it in the sidebar.

Visibility options

Click the three dots at the bottom right corner of the Items sidebar to access visibility controls:

- **Show Hidden Items** – Display all currently hidden items.
- **Invert All Item Visibility** – Swap visibility states: shown items become hidden, and hidden items become visible.

History

The History sidebar provides you with the complete feature tree timeline of all edits you make to your model. The timeline consists of individual steps that are recorded every time you make an edit. Each step represents a feature or action with an expandable card that allows you to view or edit the corresponding parameters you defined.

Explore the History sidebar

To show or hide the History sidebar, tap **History** .

Each recorded step has an expandable card to view the parameters you previously defined. To expand a history card, click/tap the arrow beside the step.

History step settings

You can hover over a step and select the three dots to access the step settings. The step settings give you the option to do the following:

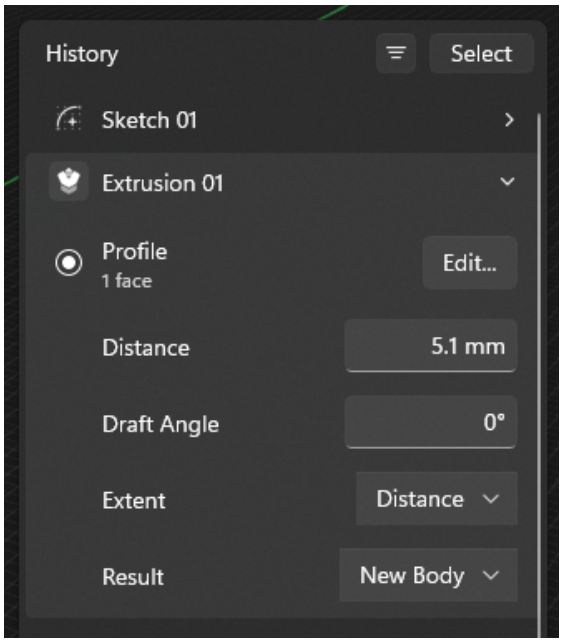
- **Insert Breakpoint** – Adds a breakpoint after the step. Steps after the breakpoint are temporarily disabled and will not be visible on your model. To remove a breakpoint, simply hover over the breakpoint and then select the X.
- **Suppress/Unsuppress** – Suppressing a step hides the step from the history and removes it from your design-build history while unsuppressing a step adds the step back to your model history. Suppressing and unsuppressing a step is useful for instances where you want to remove a step you created but don’t want to temporarily delete it.
- **Zoom To** – Zooms to the part of your model where you performed the specific step.
- **Rename** – Updates the step name.
- **Duplicate** – Duplicates the step.
- **Delete** – Deletes the step.
- **Expand/Collapse** – Another way to expand or collapse the history card.

History card parameters

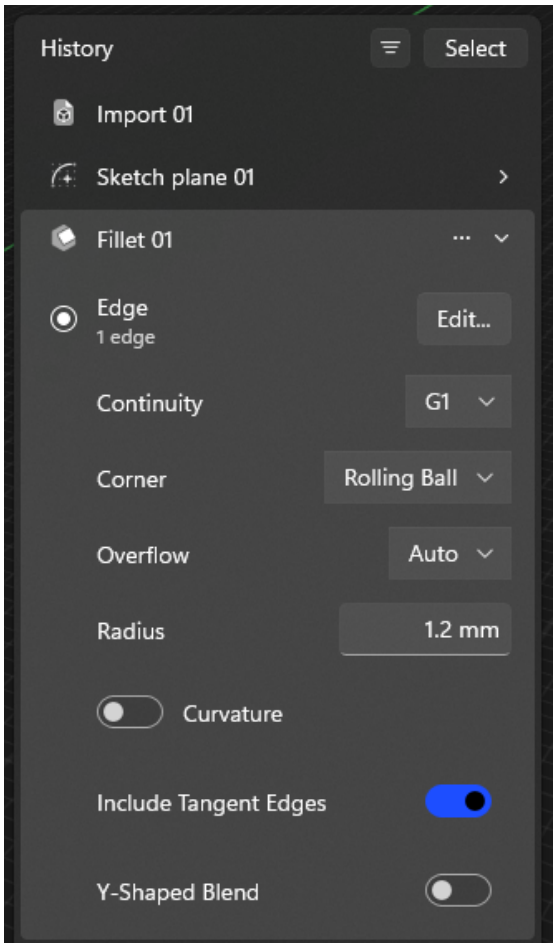
You can view or modify parameters you defined for relevant steps in their equivalent **History** card.

Below are some examples of parameters you can find in the history card for Extrude, Fillet, and, Chamfer:

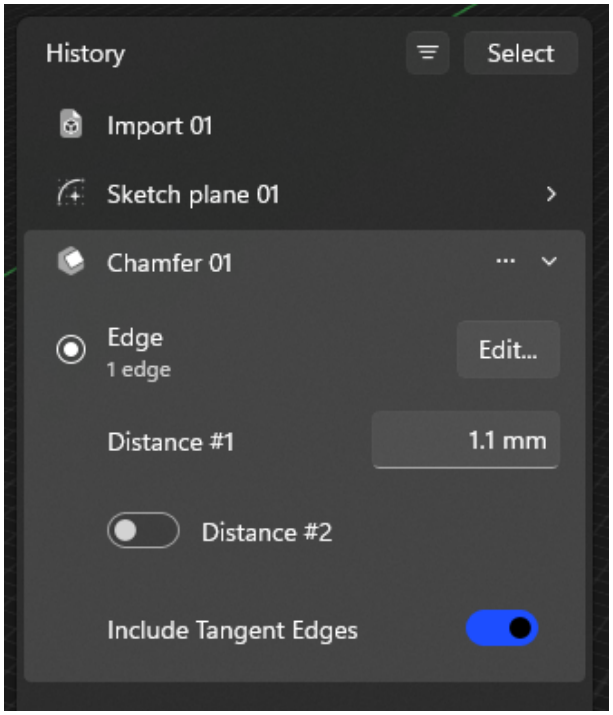
Extrude



Fillet



Chamfer



Filter History steps

With the design history, all your modeling steps are recorded so you can always get back and modify the right step. As your model gets more and more complex, finding the step to edit may take some time. To help with that, you can filter steps for isolated or selected items in the History sidebar.

Filter by: Isolated Items

When you filter by isolated items, the History sidebar automatically filters the history, so only those steps will be shown that are relevant to the isolated items.

To filter items by isolation:

1. Select any part of your model.
2. Turn on **Isolate mode**.

Filter by: Related to selection

To find a step of a certain detail on your model, simply select the detail on your model.

Once you select the detail on your model, Shapr3D will only show those steps that are relevant to your current selection. When you filter by selection, the History sidebar automatically filters out the unrelated items for your selection.

To filter items by selection, simply select any part of your model.

Merge History steps

You can merge the history steps for your model using the **Merge** option. By merging history steps:

- All your sketches unlink or break their connections from bodies.
- You combine all previous steps and delete all steps before and after the breakpoint.
- You have the option to delete or keep sketches.
- You have the option to delete or keep variables.

> **Important**
>
> You cannot recover merged history steps once you quit your project.

To merge history steps:

1. Open **History**.
2. From the History sidebar, select **History**.
3. Choose what you want to happen to your sketches after merging:
    - **Delete sketches** – Deletes your sketches after merging.
    - **Keep sketches** – Keeps your sketches after merging.
    - **Delete Variables** – Deletes your variables after merging.
    - **Keep Variables** – Keeps your variables after merging.
4. To finish, select **Merge**.

Video walkthrough

https://youtu.be/rmmizsmc1Hk

Manage projects and folders

You can access and organize your projects from two main sections in the Shapr3D Dashboard: **Recents** and **Spaces**. Use these sections to browse, open, rename, move, and manage your designs and folders.

Access your projects

You can open any project directly from the **Recents** or **Spaces** section in the Shapr3D Dashboard. Just select the project tile to load it in the modeling space.

Browse your projects

Projects are shown as tiles across **Spaces**, **Recents**, and any organized folders. These tiles give you a quick overview of key details at a glance:

- **Preview** – A thumbnail image of your model for easy identification
- **Title** – The name of the project
- **Last edited by** – The name of the last team member in a shared space who edited the design and the time of the last update
- **Storage status** – A cloud icon appears if the project is stored only in the cloud. If the icon isn't visible, the project is available locally on your device.

Manage your projects

Recent projects

The Recent Projects under Recents shows your most recently opened or modified projects across all spaces. From here, you can right-click (secondary click) a project to:

- Open the project
- Rename the project
- Manage storage location
- Show project versions
- Export the project

For more advanced actions like organizing projects into folders or restoring project versions, switch to the **Spaces** view.

Spaces

Under **Spaces**, you can find the library of your shared projects and drafts.

Create a new project

You can create a new project directly from your shared space or Drafts.

To create a new project:

1. Go to the space you want to create a project from.

2. From the top bar, select **New**  .

Import a new project

To import a project into any space:

1. Go to the space where you want to import a project.
2. From the top bar, select  or .

Manage project storage location

Choose to store your project locally or in the cloud. To learn more, visit Shapr3D Sync.

Rename a project or space

To rename a project:

1. Secondary click (right-click) or long-tap on a project or folder.
2. Select **Rename**.

To rename your space:

1. Right-click (secondary click) or long-tap on your space.
2. Select **Rename Space...**

Selecting projects

You can select projects in the Dashboard to manage them individually or in groups.

Selecting a project

To work with a single project, select it and open the context menu:

- **Windows**: Right-click on the project.
- **iPadOS**: Tap and hold the project.
- **macOS**: Secondary click on the project.

Selecting multiple projects

To manage multiple projects, select them first, then open the context menu for batch actions:

- **Windows**: Click  to select the projects, then right-click to open the context menu.
- **iPadOS**: Tap **Select** to choose the projects, then tap and hold to open the context menu.
- **macOS**: On your keyboard, press and hold **Shift** to choose the projects, then secondary click to open the context menu.

Duplicate a project

To duplicate a project:

1. Secondary click (right-click) or long-tap on a project or folder.
2. Tap **Select** or to select multiple projects and then select **Duplicate**.

Restore project versions

You can view or restore previous versions of your projects by right-clicking on a project and selecting **Show Versions**. To learn more, visit Project Versions.

Quick export to your favorite formats

Effortlessly export your projects to your favorite formats using the Quick Export option in the Dashboard. With this feature, you can select multiple projects and export them all at once, making your workflow smoother and more efficient.

To export your projects to your favorite formats:

1. Select the project(s) to export.
2. Right-click (secondary click) or long-tap on any of your selected projects.
3. Select **Quick Export to Favorite Formats.**
4. Select a folder or location to save your projects.

Export a project to .shapr file

To export a project to a .shapr file:

1. Select the project(s) to export.
2. Select Save as .shapr file.

Create and manage folders

You can create folders under any of the available spaces to organize your projects more easily. You can find your created folders in both the main Dashboard and Dashboard sidebar.

To create a new folder:

1. Select **New folder**  in the top-right.
2. When prompted, enter a folder name and then select **Create**.

To organize projects and folders into other spaces or folders, do any of the following:

- Drag and drop projects or folders from either the main Dashboard or Dashboard sidebar within or across the available spaces.
- Secondary click (right-click) on projects and folders and then select **New Folder with Selection** to automatically move them into a new folder.

Folder navigation

You can navigate between folders and subfolders using any of the following options:

- Directly from the Dashboard sidebar
- By using the navigation arrows for iPadOS and macOS
- By using the breadcrumbs for Windows to navigate

To navigate between folders:

- **Windows**: From the breadcrumb trail at the top, click a folder or subfolder name to go back to its location.
- **iPadOS/macOS**: Click or tap any of the back or next arrow keys at the top to go to the next or previous folder location.

Sort projects

To sort projects within a space, click/tap **Sort**  from the top bar and then choose a sorting type:

Windows:

- **Date modified** – Sorts projects by date.
- **Name** – Sorts projects by name.
- **Ascending** – If Date modified is selected, projects and folders sort in ascending order.
- **Descending** – If Name is selected, projects and folders sort in descending order.

iPadOS/macOS:

- **Date Modified (Latest to Oldest)** – Sorts projects by date in descending order.
- **Date Modified (Oldest to Latest)** – Sorts projects by date in ascending order.
- **Name (A to Z)** – Sorts projects and folders by name in ascending order.
- **Name (Z to A)** – Sorts projects and folders by name in descending order.

Delete a folder or project

Secondary click (right-click) or long-tap on a project or folder, or tap Select or ☑☰ to select multiple projects and then select **Delete**.

> **Important**
>
> All projects are automatically synced to the cloud. When you delete a project, it’s removed from all devices and can’t be recovered. To keep a backup, export the project before deleting.

Project Versions

The Project Versions feature in Shapr3D tracks and restores previous versions of your projects, ensuring no work is lost. Whether recovering from accidental changes or comparing different design stages, this feature provides control over your project's revisions.

Feature overview

Project Versions offer several benefits that help you manage your project revisions efficiently:

- **Project safety**: Restore previous versions of your project with ease, protecting your work from accidental changes.
- **Flexibility**: Switch between versions to compare different stages of your design process.
- **Simple navigation**: Access your version history directly from the dashboard, making it easy to manage backups.

Access Project Versions

To access a project's previous version:

1. Go to the **Dashboard**.
2. Under the **Recents** or **Spaces**, long-tap or right-click (secondary click) on the project for which you want to access the versions.
3. From the context menu, select **Show Versions** to open the Project Versions pop-up.
4. Scroll down to browse through the available versions and check the version information in the **Information** section**.**
    - **Version**: The version number in the list
    - **Copy of**: Shows the original version number if this version was restored from a previous one
    - **Edited by**: The user who made changes in that version
    - **Version** created: The date and time the version was saved
    - **Synced**: Indicates if the version is synced to Shapr3D Cloud
    - **Client**: The device or platform where the version was created or edited

Restore to a previous version

To restore to a previous version:

1. Open Project Versions for your project.
2. Browse through the available versions and select the one you want to restore as the latest.
3. To continue, select **Restore as Latest**.

Share project details to your team

Collaborate more effectively by sharing project details with your team through a simple link. Team members can view project information, download the file, or open it directly in Shapr3D from their browser.

How to share a project

1. Access the **Share this Project** dialog from any of the following:
    - From the modeling space, select **Share.**
    - From the modeling space, access the Export dialog and under **All Formats,** select **Project**.
    - From the Dashboard, right-click (secondary click) on a project and then select **Share**.
2. Select **Share** and do any of the following:
    - Select **Move to Space** to share with team members in the space you're a member of.
    - Choose to either copy the link directly or share via messaging platforms.
    - Add email addresses of individuals you want to invite as a commenter or a viewer.

What your team can do with the link

When you share a link to your project details, team members with permissions can view the project information through their browser.

Team members with permission can:

- **View the project information** – See project details and preview the design thumbnail
- **Download the design file** – Save the project as a .shapr file to their device
- **Open directly in Shapr3D** – Launch the project in Shapr3D for viewing or editing
- **Access Published Versions** – Open the Published Version if available for the project

General settings

Settings, peripherals, and accessories

Navigation Presets

You can customize how you navigate in Shapr3D by choosing a preset under **Navigation Presets**. Navigation presets define how mouse, trackpad, touch, or pen inputs control orbiting, panning, and zooming in the modeling space.

Access Navigation Presets

To access navigation presets:

Windows/macOS

1. Go to **Settings** or **Preferences**.
2. In the **General** tab, select **Navigation Presets...**
3. Select your preferred preset under **Shapr3D Presets** or choose among the available CAD software presets under **Popular Presets**.
   Note: When you select a preset, the panel on the right shows a visual breakdown of how pan, orbit, and zoom are mapped for that preset.
4. To save the changes, click **Apply** (Windows) or **Apply preset** (macOS).

iPadOS

1. From the **Dashboard** or modeling space, open **Settings**.
2. Under the Navigation section, select **Navigation Presets**, then choose between **Shapr3D Default** and **Shapr3D Classic**.
   Note: Shapr3D Default is optimized for three-button mouse navigation, while Shapr3D Classic restores the original Shapr3D navigation settings. To learn more, see Shapr3D Presets.

Shapr3D presets

Shapr3D presets are designed to match common navigation workflows across devices, including mouse, trackpad, touch, and pen input.

When you select a preset, the panel on the right shows a visual breakdown of how pan, orbit, and zoom are mapped for that preset.

Recommended

Recommended presets are optimized for Shapr3D's default navigation behavior and are a good starting point if you're new to the software or prefer a setup designed specifically for Shapr3D.

Shapr3D Default

Shapr3D Default is optimized for modern three-button mouse and trackpad workflows. Camera controls are mapped to minimize modifier key use and keep navigation consistent across input devices. This preset is recommended for new users and for anyone who prefers streamlined camera control.

Shapr3D Default navigation behavior by input device:

- **Mouse**
  Pan and orbit are separated into distinct mouse button actions, with zoom handled by the scroll wheel. Modifier keys are used only when needed for alternate navigation.
- **Magic Mouse**
  Touch-based gestures are mapped to mirror mouse behavior, keeping pan and orbit predictable while maintaining smooth zoom control.
- **Trackpad**
  Two-finger gestures are used for pan and orbit, with zoom handled through vertical gesture movement, reducing the need for keyboard modifiers.

Shapr3D Classic

Shapr3D Classic restores the original navigation behavior from earlier versions of Shapr3D. It preserves legacy mappings for pan, orbit, and zoom, which may feel more familiar to long-time users.

Shapr3D Classic navigation behavior by input device:

- **Mouse**
  Pan, orbit, and zoom rely more heavily on modifier keys, following the original Shapr3D interaction model.
- **Magic Mouse**
  Gesture mappings follow the legacy behavior, maintaining consistency with earlier Shapr3D releases.
- **Trackpad**
  Touch gestures reflect the original navigation logic, with modifier keys used more frequently to switch between camera actions.

Popular presets (Windows and macOS only)

Popular presets help you get productive faster by matching the navigation behavior of other widely used CAD and 3D design tools. These presets are available on Windows and macOS and are useful if you're transitioning to Shapr3D from another application.

- Alias
- CATIA
- NX
- Blender
- Plasticity
- Fusion 360
- OnShape
- Rhino
- SketchUp
- SolidWorks

Settings or Preferences

Settings (Windows & iPadOS) or **Preferences** (macOS) is your central hub for configuring the Shapr3D environment. This menu allows you to:

- **Manage your Account** – View your login email, manage subscriptions, or log out.
- **Configure Sync** – Monitor project sync status and trigger manual syncs.
- **Customize the Interface** – Set your preferred language, theme (Light/Dark), accent colors, and menu placement.
- **Adjust Graphics & Performance** – Fine-tune resolution, anti-aliasing, and tessellation for your hardware.
- **Set Navigation Styles** – Select navigation presets and toggle scroll directions to match your workflow.
- **Check App Information** – Verify your specific software version and installation type.

Access Settings or Preferences

- **Windows**:
  - From the **Dashboard,** open the **Team Switcher** > **Settings.**
  - From the menu bar, go to **File** > **Settings.**
- **iPadOS**:
  - From the **Dashboard,** open the **Team Switcher** > **Settings.**
  - From the modeling space, open **Settings** > **Settings**.
- **macOS**:
  - From the **Dashboard,** open the **Team Switcher** > **Settings.**
  - Go to the Mac menu bar > **Shapr3D** > **Settings...**

Shapr3D settings

> **Tip**
>
> The setting location or placement may vary depending on your device.

Account settings

- **Email Address** – Shows the email address logged in to Shapr3D.
- **Log Out** – Logs you out of Shapr3D.
- **Manage Subscription** – Redirects you to the Shapr3D Subscription page in your browser.
- **Restore Subscription** – Restores your previous subscription.

General settings

- **Language** – Choose your preferred supported language.
- **Theme** – Choose your preferred theme: **System**, **Light**, **Dark**.
  - **Always** – Menu labels are always displayed.
  - **On Hover, On All Buttons** – The menu labels only display upon hovering your pointer.

    - **Only for Hovered Buttons** (Windows) – Only the hovered button displays the menu label.
- **Accent color** – Choose your preferred accent color.
- **Undo/Redo Button Position** (Windows) – Choose **Title Bar** or **Bottom Bar** as your preferred placement for the **Undo** and **Redo** buttons.
- **Interface** (iPadOS) – Choose between **Left** or **Right** to switch the placement of the menus and tools to the left-hand or right-hand side of your screen.
- **Single Key Action**
    - **Command Search** – Uses Command Search as your single key action.
    - **Hotkeys** – Uses Hotkeys as your single key action.
    - **Customize keyboard shortcuts** – Opens the **Keyboard Shortcuts** settings.
- **Keyboard Shortcuts** (macOS) – Select **Customize…** to open the keyboard shortcuts settings.
- **Selection Extension** (macOS) – Choose whether you need to hold Shift to select multiple items. When disabled, each click adds to your selection automatically.
- **Rendering Quality** (macOS) or **Resolution** (Windows) – Controls the overall visual fidelity when displaying models in both Modeling and Visualization spaces. Choose **High**, **Medium**, or **Low** to balance image quality and performance.
- **Tessellation Quality** – Adjusts how detailed mesh surfaces appear when displayed or exported. Choose from **Very High**, **High (default)**, or **Low** to balance visual smoothness and performance.
- **Circular Annotations** – Choose between **Always Radius** or **Radius and Diameter** as your preferred annotation for circular sketches and geometry.
  Note for Windows: You can change your circular annotation settings from Views and Appearance.

Import

- **Show advanced preferences every time** – Enable this for faster processing of STEP, IGES, CATIA Part, SolidWorks Part, and Solid Edge Part files. To learn more about these settings, visit Import.

Graphics

- **Anti-Aliasing** (Windows) – Control how edges are smoothed in the workspace. Choose from **Disabled**, **2× MSAA**, or **4× MSAA** to balance visual quality and performance.
- **Rendering Quality** (macOS) or **Resolution** (Windows) – Controls the overall visual fidelity when displaying models in both Modeling and Visualization spaces. Choose **High**, **Medium**, or **Low** to balance image quality and performance.
- **Tessellation Quality** – Adjusts how detailed mesh surfaces appear when displayed or exported. Choose from **Very High**, **High (default)**, or **Low** to balance visual smoothness and performance.

Navigation settings

- **Navigation Presets** – Opens **Navigation Presets**.
- **2-Finger Rotation** (iPadOS) – Toggle to enable or disable 2-Finger Rotation.
- **Zoom** or **Reverse Zoom Scroll Direction** – Toggle to enable reverse zoom scroll direction.
  Note: If you're using a keyboard, visit **Navigation Presets** for more navigation controls.
- **Start Shapr3D with** – Select the screen that appears when you launch Shapr3D: **Recents**, **Projects**.
- **Tutorial mode** (macOS and Windows)
    - **Display clicks** – Enable to display pointer clicks while modeling.
    - **Display keyboard presses** – Enable to display keyboard presses while modeling.

Note: Refer to **Other settings** below to learn how to use tutorial mode on iPadOS.

Sync settings

- **Sync Projects** – Shows you the status of when your project was last synced. You can select this option to manually sync your projects.

About

- **App Version** – Check your software version.
- **Terms of Use** – Access the Shapr3D Terms of Use.
- **Privacy Policy** – Access the Shapr3D Privacy Policy.
- **Data Setting** – Open to enable or disable **Send Usage Data**.

Account Deletion settings

- **Delete Account and Projects** – Deletes your account and projects.
  Warning: This action cannot be undone. Once prompted, you will lose every project you worked on in Shapr3D.

Other settings (iPadOS only)

For iPaOS, open the system Settings, and find Shapr3D in the list of installed applications to access these additional settings:

- **Allow Shapr3D to Access** – Enables access to specific device settings such as photos and camera.
- **Preferred Language** – Another way to choose your preferred supported language.
- **Tutorial Mode** – Enables **Display Pencil and touches** and **Display keyboard presses**. Restart Shapr3D for these changes to apply.

Shapr3D supported languages

Shapr3D is available in multiple languages, with the entire app interface fully localized so you can design in the language that feels most natural to you.

List of supported languages in Shapr3D

The Shapr3D app user interface is fully localized for the following languages:

- English
- Chinese, Simplified (简体中文)
- Chinese, Traditional (繁體中文)
- German (Deutsch)
- Spanish (Español)
- French (Français)
- Italian (Italiano)
- Japanese (日本語)
- Korean (한국어)
- Portuguese, Brazil (Português, Brasil)
- Russian (Русский)

How to change language settings in Shapr3D

You can change language preferences directly within the Shapr3D app interface, either through Settings or Preferences when logged in, or from the Home Page when logged out.

Change language via Settings or Preferences (Logged in)

1. Navigate to the **Settings** or **Preferences**.
2. Select **Language** to open the dropdown.
3. Select your preferred default language choice from the dropdown list.
4. To save the changes, select **Done**.

Change language from the Shapr3D app home page (Logged out)

1. Open Shapr3D to launch the homepage.
2. Select the language at the bottom to open the language menu.
3. Select your preferred language to instantly update the default language.

Peripherals and accessories

You can use peripherals and accessories compatible with Shapr3D to enhance your modeling experience.

Explore the list of compatible peripherals and accessories below to learn how to set up and configure them.

- SpaceMouse
- Wacom pen tablets and display

SpaceMouse

SpaceMouse by 3Dconnexion allows you to navigate effectively in 3D space and run various commands in Shapr3D.

Windows

To use SpaceMouse with your device:

1. Go to 3Dconnexion's website and download the latest version of 3Dconnexion's driver, 3DxWare64 or 3DxWare32.
2. Install the driver.
3. Once the 3Dconnexion Home program automatically opens after installing the driver, plug your SpaceMouse into the computer via the USB cable.
   Note: If you're using the wireless SpaceMouse, connect the dedicated receiver and launch the pairing tool in the 3Dconnexion Home program. For more information on pairing the wireless 3D mouse, visit How do I pair a dedicated receiver with my wireless SpaceMouse?.
4. Verify that the device works correctly using the test application installed by the driver.
5. **Optional:** While using Shapr3D, you can change the navigation mode of your SpaceMouse in the **Views** tab under **Views and Appearance**.

   Note: For more information on SpaceMouse navigation modes, refer to the SpaceMouse manual.

macOS

To use SpaceMouse with your device:

1. Go to 3Dconnexion's website and download the latest version of 3Dconnexion's driver.
2. Install the driver.
3. Once the 3Dconnexion Home program automatically opens after installing the driver, restart your Mac.
4. Plug your SpaceMouse into the computer via the USB cable.
   Note: If you're using the wireless SpaceMouse, connect the dedicated receiver and launch the pairing tool in the 3Dconnexion Home program. For more information on pairing the wireless 3D mouse, visit How do I pair a dedicated receiver with my wireless SpaceMouse?.
5. Allow the program to control your Mac in any of the following ways:
   - **From the Accessibility Access dialog** – If this dialog automatically appears after connecting SpaceMouse to your computer, click **Open System Preferences**, then select **3DconnexionHelper**.
   - **From the Mac System Settings** – Go to **System Settings** > **Privacy & Security** > **Accessibility**. Look for **3DconnexionHelper**, then enable it to allow access.

Note: To learn more about Mac system settings, visit Allow accessibility apps to access your Mac.

6. You can reconfigure the mapping and view command names for Shapr3D by going to your Mac **System Settings > 3Dconnexion**, or opening **3Dconnexion Home** then selecting **Properties**.

Wacom pen tablets and displays

The compatibility of Shapr3D with digital pen input makes it a great match for Wacom devices.

Choosing your Wacom device

Wacom devices can be split into the following two categories:

- **Pen displays** feature a built-in screen, and are sold under the brand names "*Wacom One*" and "*Cintiq.*"
- **Pen tablets** are screenless, and are sold under brand names such as "*One by Wacom*" and "*Intuos.*"

There are advantages and disadvantages to both. *Pen displays* are less portable and more expensive, while using a *pen tablet* takes some getting used to, as you need to look at a separate screen.

Shapr3D works with both types of devices, but **we recommend devices with a built-in screen**, as they are more intuitive to use.

Touch or no touch

Wacom sells both *pen displays* and screen-less *pen tablets* with multi-touch support in addition to pen input.

At the moment, we only support multi-touch input for *view navigation* on *pen displays*, but not on *pen tablets.* While you can rely on the built-in and customizable shortcuts that the Wacom driver provides for pen tablets, you can't navigate the 3D space with touch input as you would on iPad.

For the best experience, **we recommend touch-enabled pen displays**, but Shapr3D works without touch input as well.

Which pen to use

You can get started with the pen that was shipped with your Wacom device. For the best experience, a pen specifically designed for 3D applications is recommended, such as the *Pro Pen 3D*. Notably, its additional button allows you to navigate in 3D space using your pen only, which many users find more convenient than using keyboard shortcuts together with pen gestures.

Navigation

If you have a touch-enabled pen display, you can navigate in 3D space using touch gestures, the same way you'd navigate on an iPadOS or a Windows device with a built-in touchscreen. If you have a screenless pen tablet or a pen display that has no touch input, view navigation happens with the pen.

Navigation with keyboard

With the default settings, you can navigate the 3D space using a combination of keypresses and either hovering your pen over the tablet, or touching and dragging it over the surface. You can use the following modifiers:

- Shift: Tumble (also called Orbit)
- Ctrl: Pan
- Alt: Zoom in 3D space
- Space: Focus the view on the face under the pointer

Note: The Shapr3D onboarding tutorial teaches you these gestures when you get started. To revisit the tutorial, go to the **Dashboard** > **Tutorials** > **Redo the Basics**.

To learn more about pen navigation and how to configure your Wacom device, visit this Setting Up Your Wacom Tablet for Shapr3D Modeling.

Views and snapping

Views, appearance, and snapping options
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Views and Appearance

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You can access the **View** and **Appearance**  tabs to update your preferred view and appearance settings for your modeling space such as grid position, projection, and surface analysis, and save up to eight orientation views for quick access.

To access the view and appearance settings:

- **Windows**: Select  to access the **Views** and **Appearance** tab settings or select **View** from the menu bar.
- **iPadOS**: Select  to access the **Views** and **Appearance** tab settings.
- **macOS**: Select  to access the **Views** and **Appearance** tab settings or select **View** from the menu bar.

Views

- **Default View** – Select to reset your view.
- **Views** – Select between **Top**, **Bottom**, **Front**, **Back**, **Right**, **Left**, and **Nearest Ortho View** as your preferred plane during 2D modeling.
  Note: You can also use the Orientation Cube to choose a plane or reset your view.
- **Saved views** – Save up to eight of your orientation views to easily get back to a specific view of your mode.
  - To save a view, hover over any of the available views and then select **Save**.
  - To update or replace a saved view, right-click (secondary click) on the view you want to replace and then select **Update**.
  - To delete a view, right-click (secondary click) on the view you want to replace and then select **Delete**.
- **Grid Position** – Choose between **XY Plane**, **YZ Plane**, and **ZX Plane** as your preferred planar orientation.

Appearance

- **Animate Camera** – Toggle on or off to enable smooth transitions between views.
- **2-Finger Rotation (Touch and pen)** – Enable this option to use two fingers to rotate your view without moving the position of the camera.
- **Field of view (Orthographic vs. Perspective)** – Adjust the camera angle between orthographic (0°) and perspective (90°) using the slider.
- **Show Hidden Edges** – Turn on to highlight and select edges hidden behind bodies. This makes bodies transparent, helping you inspect interior geometry.
- **Show Pinned Measurements** – Enable to always display pinned measurements in the design space.
- **Circular Annotations (Windows only)** – Choose between **Always Radius** or **Radius and Diameter** as your preferred annotation for circular sketches and geometry.
  Note: To change circular annotations for iPadOS and macOS devices, go to Settings or Preferences.

Additional View options for desktop

For macOS and Windows, you can access additional view options from the **View** menu in the menu bar. The additional options available depend on your device and the space you are currently in.

- **Show Properties Sidebar** – Show or hide sidebar properties for the space you are currently in
  - **Modeling space** – Shows History sidebar
  - **Visualization space** – Shows Properties sidebar for Visualization settings
  - **Drawings space** – Shows Properties.
- **Material Properties** – Select to open the Material tab when in Visualization space.
- **Environment Properties** – Select to open the Environment tab when in Visualization space.
- **Camera Properties** – Select to open the Camera tab when in Visualization space.
- **Drawing Properties** – Select to open the Drawing properties sidebar when in Drawings space
- **Go to Modeling** – Go back to modeling space when in Visualization or Drawing space.
- **Collapse All** – Collapses the expanded items in the Items Manager and steps in the History sidebar.
- **Rotate View** – Rotates the camera view in fixed angle increments.
- **SpaceMouse Settings** – Opens settings for a connected SpaceMouse.
- **Show Items Sidebar** – Shows or hides Items sidebar.
- **Show Measurements** – Turns on or turns off Measure mode.
- **Enter Full Screen** – Expands Shapr3D to full screen.

Orientation Cube

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The **Orientation Cube** is a tool that lets you manually navigate around the modeling space while giving you a real-time view of your current orientation. Each face of the cube represents the sides of the planar surfaces which makes it useful for 2D modeling.

Design interface

The design of the **Orientation Cube** consists of the following:

- **Axes** – Has X, Y, and Z axes identical to the modeling space.
- **Face** – There are six faces: **Top**, **Bottom**, **Front**, **Back**, **Right**, and **Left**.
- **Corners** – There are eight corners:
    - Top/Front/Right
    - Top/Right/Back
    - Top/Back/Left
    - Top/Left/Front
    - Front/Right/Bottom
    - Right/Back/Bottom
    - Back/Left/Bottom
    - Left/Front/Bottom
- **Edges** – There are 12 edges:
    - Top/Front
    - Top/Right
    - Top/Back
    - Top/Left
    - Right/Back
    - Back/Left
    - Left/Front
    - Front/Right
    - Front/Bottom
    - Right/Bottom
    - Back/Bottom
    - Left/Bottom
- **Rotation arrows** – In planar view, there are rotation arrows above the top edges of the cube that rotate your plane clockwise or counterclockwise.

Navigation

Set to default view

- **Keyboard and mouse/trackpad** – Double-click the cube to reset it to the default view.
- **Touch and pen** – Using your finger or pen, double-tap the cube to reset it to the default view.

Rotate the cube

- **Keyboard and mouse/trackpad**: Click and drag the cube to rotate it.
- **Touch and pen:** Use your finger or pen to drag the cube to rotate it.

2D view

Select any face of the cube to go into the 2D view of that specific planar surface.

To rotate the plane in a clockwise or counterclockwise manner while in 2D view, select the left or right rotation arrows.

Orthographic view

Select an edge or corner of the cube to go into an orthographic view for that specific edge or corner.

Views and Appearance

Access the Views and Appearance settings for more advanced options to navigate the modeling space.

Snapping Options

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Shapr3D sketch tools automatically snap to elements in your project and graphical elements in the user interface, such as the grid. You can adjust settings in **Snapping Options** to modify this behavior as desired.

Snap To settings

Shapr3D helps you define precise geometries by allowing you to snap to important points. These just offer suggestions: you can always define points between the snap points. You can control the snapping behavior of the following:

- **Grid**  – Snap to points on the grid displayed for the active plane.
- **Sketch Guidelines**  – Sketch guidelines are displayed as purple extensions from sketch elements. They help you identify where you can achieve constraints such as coincident and tangent.
- **Sketch Guidepoints**  – Sketch guidepoints include endpoints, midpoints, arc center points, and profile center points.
- **3D Guidepoints**  – 3D guidepoints are points from a 3D body. These include vertices, edges, edge midpoints, and face and hole centers.
- **Distant Edges**  – Reference geometry outside the active sketch plane when in an orthogonal view. These elements are projected into the sketch plane to help you maintain precise alignment and constraints.

Show settings

In **Snapping Options**, the **Show** settings control the visibility of guide points in the modeling space and allow you to turn on **Snapping Hints**.

Snapping Hints provide you with visible text next to the pointer that indicates what you are snapping to.

Shortcuts

Shortcuts, gestures, and UI
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Keyboard shortcuts, gestures, and hotkeys

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You can navigate, edit, sketch, and model intuitively with your preferred input device in Shapr3D. On compatible hardware, using touch and pen activates basic gestures while a keyboard allows you to use hotkeys or do quick command searches. Follow the guidelines below to learn how to use your different input devices to interact with Shapr3D.

Getting started

Windows

Shapr3D on Windows supports simultaneous use of all input methods without any additional configuration. Some elements of the UI slightly change depending on the input method you are using, but no manual configuration is required.

iPadOS

To get started using a keyboard with a mouse or trackpad,

- connect your compatible Bluetooth devices to your iPadOS device, or
- connect your Magic Keyboard or Smart Keyboard (Folio) to your iPad.

  Note: If your keyboard doesn't have an **Escape** key and you need one for setup or other functions you can assign another key (like the Globe key) to act as **Escape**. You can do this in your iPad's Settings under **General** > **Keyboard** > **Hardware Keyboard** > **Modifier Keys**. To learn more, visit Apple's support page: Use shortcuts on an Apple external keyboard with iPad.

If you're using a mouse,

- enable secondary click (right-click) to make sure you have full mouse capability.

If you're using a trackpad,

- you can register taps as clicks if you prefer to tap instead of press.

macOS

Shapr3D on macOS supports keyboard and mouse usage. No manual configuration is required.

Single Key Action setting

You can choose between two **Single Key Action** settings, **Hotkeys** and **Command Search,** to quickly access commands and tools via the keyboard.

Hotkeys

Hotkeys are the default Single Key Action setting. When prioritizing hotkeys, you can select tools and commands with a single key press or a simple key combination.

On new installations of Shapr3D, hotkeys are enabled by default and accessing command search requires an extra hotkey (X).

Note: If you have an older version of Shapr3D, your default setting might still be set to trigger command search as you start to type.

Command Search

When prioritizing command search, you can start typing the name of the command and select from the search box that pops up. This action requires more keystrokes but can be easier to learn.

Choose a Single Key Action

To choose between Hotkeys and Command Search:

1. Go to **Settings or Preferences** > **Single Key Action.**
2. Choose your preferred action.

Customize keyboard shortcuts

You can customize your keyboard shortcuts to modify and assign shortcuts to the actions you use most, allowing you to streamline your workflow and enhance productivity.

To customize keyboard shortcuts:

1. Go to **Settings** or **Preferences**.
2. Open the keyboard shortcuts settings:
    - **Windows/iPadOS**: Select **Customize Keyboard Shortcuts…**
    - **macOS**: Under **General**, find **Keyboard shortcuts** and then select **Customize…**
3. Choose the category where you want to create or modify a keyboard shortcut.
4. Find the specific action, then press the keys you want to assign as the shortcut to auto-populate the field.
5. To finish, select **Save Changes**.

Cheat sheet

Below is a cheat sheet of hotkeys, shortcuts, and basic gestures for various input devices. You can also find this cheat sheet in the Shapr3D modeling space:

- **Windows**: Select **Help** > **Keyboard Shortcuts**, or long-press the **CTRL** key.
- **iPadOS**: Select **Help** > **Keyboard Shortcuts,** or long-press the **Command** key.
- **macOS**: Select **Help** > **Keyboard Shortcuts**.

Keyboard and mouse/trackpad

> ### Tip
>
> Hotkeys are highlighted in blue. To use hotkeys, choose **Hotkeys** as your **Single Key Action**.

Tool/Action	Gesture/Shortcut/Hotkey
Navigation	
Orbit (Rotate camera)	**Mouse/Magic Mouse**: Secondary click (right-click) and drag the mouse **Trackpad:** • **iPadOS/macOS:** Pan with two fingers on the trackpad. • **Windows:** Hold down **CTRL + Shift** and pan one finger on the trackpad. **Note**: These controls use the **Shapr3D Default** navigation preset. To learn more, go to Navigation presets.
Pan (Move camera)	**Mouse:** Hold down the **middle mouse button** and drag the mouse. Alternatively, hold the **Shift** key and use secondary click (right-click) to drag the mouse. **Magic Mouse**: Hold the **Shift** key and use secondary click (right-click) to drag the mouse. **Trackpad**: • **iPadOS/macOS**: Hold down the **Shift** key and pan with two fingers on the trackpad. • **Windows**: Hold down **CTRL + ALT** and pan one finger on the trackpad. **Note**: These controls use the **Shapr3D Default** navigation preset. To learn more, go to Navigation presets.
Zoom camera	**Mouse**: Scroll with the mouse wheel **Magic Mouse**: Hold the **CTRL** key and pan up/down with your finger on the mouse. **Trackpad**: Pinch in or out with two fingers on the trackpad

Tool/Action	Gesture/Shortcut/Hotkey
	Note: These controls use the **Shapr3D Default** navigation preset. To learn more, go to Navigation presets.
Zoom to face	Hover over a face and press the Space bar
Rotate 2D view by 90 degrees	Click the right or left arrow above the **Orientation Cube**.

	iPadOS/macOS	Windows
Reset view	⌘ 1	CTRL +1
Switch to front view	⌘ 2	CTRL +2
Switch to back view	⌘ 3	CTRL +3
Switch to top view	⌘ 4	CTRL +4
Switch to bottom view	⌘ 5	CTRL +5
Switch to right view	⌘ 6	CTRL +6
Switch to left view	⌘ 7	CTRL + 7

Sketching

Start sketch	Click anywhere in the modeling space
Remove last-placed point and finish sketch	**Escape** or **Backspace**⌫ key
Set final endpoint and finish sketch	**Enter** key
Add fit point to spline	Long-click anywhere on the spline

Sketch Tools		Constraint Settings	
Arc	A	Parallel	⇧ Shift + A
Circle	C	Concentric	⇧ Shift + C
Polygon	G	Equal	⇧ Shift + E
Spline	I	Horizontal/Vertical	⇧ Shift + V
Line	L	Coincident	⇧ Shift + N
Offset Edge	O	Lock/Unlock	⇧ Shift + L
Rectangle	R	Midpoint	⇧ Shift + M
Trim	T	Perpendicular	⇧ Shift + P
		Symmetry	⇧ Shift + S
		Tangent	⇧ Shift + T

Modeling

Tools		Boolean Operations	
Extrude	E	Union	⌘ U or CTRL + U
Fillet/Chamfer	F	Subtract	⌘ B or CTRL +B
Shell	H	Intersect	⌘ I or CTRL + I
Move/Rotate	M		
Translate	N		
Project	P		
Scale	S		
Revolve	V		
Sweep	W		

Edit

	iPadOS/macOS	Windows

Tool/Action	Gesture/Shortcut/Hotkey	
Search for commands	X (if hotkeys are enabled) or ⌘ F	X (if hotkeys are enabled) or CTRL + F
Undo	⌘ Z	CTRL + Z
Redo	⇧ ⌘ Z	Shift + CTRL + Z
Delete	⌫	**Backspace** or **Delete** key
Finish sketch	⏎	**Enter** key
Selection		
Select full object	Double-click the object	
Select all	⌘ A or CTRL + A	
Deselect all	**iPadOS/macOS**: Esc or ⌘ . **Windows**: CTRL + Shift + A or Esc	
Select single item	Click the item.	
Select multiple items	Hold down the ⇧ **Shift** key and click each item.	
Select area	Click anywhere in the modeling space and drag the arrow to the right or left.	
Cycle through selection filters	⇥ (**Tab** key)	
Filter selection for bodies only	B	
Filter selection for edges only	E	
Filter selection for faces only	F	
2D Drawings		
Update 2D Drawing	**iPadOS/macOS**: ⌘ R **Windows**: CTRL + R	
Project Sidebar		
Show/hide Project Sidebar	**iPadOS/macOS**: ⇧⌘ \ **Windows**: CTRL + Shift + \	
Jump to last visited document	**macOS**: ⇧⌘ [OR ⇧⌘] **Windows**: CTRL +Tab	
Close	**Esc** key	
Show/Hide Items	**iPadOS/macOS**: ⌘ ⌥ S **Windows**: CTRL + ALT + S	
Show/Hide History	**iPadOS/macOS**: ⌘ ⌥ H **Windows**: CTRL + ALT + H	
Items Manager		
Select all items/folders	**iPadOS/macOS**: ⌘ A **Windows**: CTRL + A	
Collapse all expanded folders	**macOS**: ⌥ L **Windows**: CTRL + ALT + L	
Unfocus or deselect items/folders	**Esc** key	
Delete selected items/folders	**iPadOS/macOS**: ⌘ ⌫ **Windows**: **Delete** key	
History Sidebar		
Select all steps	**iPadOS/macOS**: ⌘ A	

Tool/Action	Gesture/Shortcut/Hotkey
	Windows: CTRL + A
Collapse all steps	**macOS**: ⌥ L **Windows**: CTRL + ALT + L
Suppress selected steps	**iPadOS/macOS**: ⌫ **Windows**: **Delete** or **Backspace** key
Unfocus or deselect steps	**Esc** key
Delete selected steps	**iPadOS/macOS**: ⌘ ⌫ **Windows**: **Shift** + **Delete** key
Variables	
Create variables	**iPadOS/macOS**: ⇧⌘ V **Windows**: CTRL + Shift + V
Screenshots	
Copy to clipboard	**iPadOS/macOS**: ⇧⌘ C **Windows**: CTRL + Shift + C

Touch and pen

Tool/Action	Gesture/Shortcut/Hotkey
Navigation	
Pan (Move camera)	**iPadOS**: Pan with two fingers on the screen, **Windows**: Hold down the **CTRL** key and drag or hover over the tablet with your pen.
Orbit (Rotate camera)	**iPadOS**: Tap the grid with one finger and drag on the screen. **Windows**: Tap the grid with one finger and drag on the screen, or hold down the **Shift** key and drag or hover over the tablet with your pen. **Note**: If you enable 2-Finger Rotation, you can also use two fingers in a twisting motion to rotate the view.
Zoom camera	**iPadOS**: Pinch in or out with two fingers on the screen. **Windows**: Hold down the **Alt** key and drag or hover over the tablet with your pen.
Zoom to face	**iPadOS**: Double-tap a face with your finger. **Windows**: Double-tap a face with your finger, or hover over a face with your pen and press the Space bar.
Rotate 2D view by 90 degrees	Rotate with two fingers.
Reset view	Double-tap the Orientation Cube.
Switch to front view	Tap the front face of the Orientation Cube.
Switch to back view	Tap the back face of Orientation Cube.
Switch to top view	Tap the top face of the Orientation Cube.
Switch to bottom view	Tap the bottom face of the Orientation Cube.
Switch to right view	Tap the right face of the Orientation Cube.
Switch to left view	Tap the left face of the Orientation Cube.
Sketching	
Start sketch	Use your pen to start drawing.
Finish sketch	Tap on an empty area of the grid.
Remove last-placed point	Tap the endpoint and then **Delete** in the menu.
Add fit point to spline	**iPadOS:** Long-tap anywhere on the spline.
Edit	
Search for Commands	**macOS:** Select **Search Commands** **Windows:** Select **Help** > **Search Commands**
Undo	**iPadOS**: Swipe left with three fingers or use the **Undo** icon on the screen **Windows**: Uuse the **Undo** icon on the screen.
Redo	**iPadOS**: Swipe right with three fingers or use the **Redo** icon on the screen **Windows**: use the **Redo** icon on the screen.
Delete	Tap once and tap **Delete** in the menu.
Finish sketch	Lift the pencil from the screen.

Tool/Action	Gesture/Shortcut/Hotkey
Selection	
Select wholefull object	Tap twice with the pen.
Select all	Tap all items in the **Items Manager.**
Deselect all	Tap the empty space on the grid with the pen.
Select single item	Tap once with the pen.
Select multiple items	Tap the items, one by one, with the pen.
Select area	Long-tap the screen and drag the pen across the space.

Adaptive user interface

Shapr3D has an adaptive user interface functionality that automatically recommends tools as you make selections in your projects. These tailored menus allow you to easily access everything you need without having to browse through the default menus. Taking advantage of the adaptive user interface can accelerate your workflow twice or thrice as much.

To use the adaptive user interface, pre-select elements in your project and then choose the tool you need from the recommended tools.

Below are some examples of tools that get recommended based on your selections.

Select a face	**Offset Face**  becomes active
Select a sketch profile	**Extrude**  becomes active
Select a body	**Move/Rotate**  becomes active
Select a sketch line or curve	**Sketch mode**  becomes active
Select a line and a face	**Rotate Around Axis**  becomes active.
Select a sketch profile and an axis	**Revolve**  becomes active.

In the example below, we select a body, and the **Move/Rotate** tool becomes active in the adaptive menu. From here, the body can be moved using the gizmo without having to interact with the menus.

As you continue to make selections, Shapr3D will further recommend tools.

For example, if you select two faces from different bodies, the adaptive menu provides access to tools such as Align and Replace Face.

Tip

If you do not see the tool you are looking for directly in the adaptive menu, select **More** ° ° ° to access additional tools that are valid for your selections.

By taking advantage of the adaptive user interface, you can complete tasks without interacting with menus, significantly increasing your efficiency. You can also complete many of the tools accessed from the adaptive menu without having to select **Done**. To do this, simply select an empty area of the grid.

Read more

For detailed information about making selections in your projects, visit Selecting geometry.

Context menus

When modeling with a keyboard and mouse/trackpad, you can use context menus to select commands, actions, or options relevant to your workflow.

Context menus are available for the following features:

- **Orientation cube** – Provides relevant options such as the **Default View**, **Top View**, and **Zoom to Fit**.
- **Sketching** – Provides relevant options to the sketching tool you selected such as the option to add a control point spline after you select **Spline**.
- **Modeling** – Provides relevant options to 3D modeling such as **Isolate selection** and **Zoom to Selection**.
- **Visualization** – Provides common options for Visualization such as **Set as Focus Point** and **Remove Focus Point**
- **2D Drawings** – Provides 2D drawing-related options such as **New Font View** and **Add Bodies**

To open a context menu, simply use your secondary click (right-click), and then select your preferred action.

Read more

To learn about other quick methods to access commands, actions, or options, visit Keyboard shortcuts, gestures, and hotkeys.

Command Search

You can quickly access any command using the command search function. **Command Search** allows you to type the name of the tool you are looking for and activate it from the search results. Searching for commands can save you the time you might otherwise require to browse through menus.

Note: For information on other options for accessing tools, see **Accessing tools**.

Accessing Command Search

To learn how to access the **Command Search** function, expand the sections below:

Keyboard and mouse/trackpad

There are several ways to access **Command Search**:

- Hotkey or keyboard shortcut:
  - **macOS**: Press **X** or **Command + F**
  - **Windows**: Press **X** or **Ctrl+F**
- Use the top bar (macOS or Windows only)
  - **Windows**: Select the **Magnifying Glass**
  - **macOS**: Type in the **Search Commands** field
- From the main menu, select **Search Commands**

> **Read more**
>
> If you have an older version of Shapr3D, your default setting might still be set to trigger command search as you start to type. If you are on this setting, you can't use hotkeys or keyboard shortcuts for actions. For more details, visit Keyboard shortcuts, gestures, and hotkeys.

Touch and pen

1. Go to **Help** > **Search Commands**.
2. Display the on-screen keyboard.
3. Use the on-screen keyboard to begin typing the name of a command.

Tips for Command Search

The search box automatically updates a list of search suggestions as you type. You can launch tools directly from the search field or the list of suggestions. You can also take advantage of other functionality of **Command Search**, like using it to access recent tools or pre-selecting items in your project to initiate a search for selection-based results.

> ### Tip
>
> You don’t need to be precise with your search terms: Shapr3D automatically finds the best match to the letters you typed so you can even use shortcuts like “p3” for “Add Plane - 3 Points” or “snu” for “Scale - Non-uniform”.

Explore the table below for more information on how to utilize these functions.

	Touch and pen	Keyboard and mouse/trackpad
Auto-complete text		Press the right arrow
Cycle through suggestions		Press the up and down arrows
Run a command from search suggestions	Select a suggested command in the list	When a suggested command is highlighted, press **Return** or **Enter** Click a suggested command in the list
Clear the search	Tap the **X** icon	Press **Delete** Click the **X** icon
Access recent commands	Clear any text from the search field	Clear any text from the search field
Search results based on a selection	Pre-select items, then launch Command Search	Pre-select items, then launch Command Search
Close Command Search	Tap outside of the search box	Press **Escape** Click outside of the search box

Shapr3D modeling

Controls, modes, menus, and tools
[Section link | Table of contents | Back | Next]

Basic controls and access

Controls, selection, accessing tools
[Subsection link | Table of contents | Next]

Selecting geometry

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You can use a pen, mouse, or trackpad to select any geometry. There are various methods available for selecting geometry based on your input device and what you need to select.

Menus are customized based on your selection in the modeling space. For example, when you select a body, tools appropriate for working with bodies become available in the menu. Or when you select a sketch element, the sketch tools are automatically made available. For more information, visit Accessing tools.

> **Tip**
>
> Fully-defined sketches appear in green, while under-defined sketches appear in blue. If a sketch is fully defined, it will remain fixed unless a constraint or dimension is removed.

Basic selection

Basic selection can be performed via the Items Manager or using simple keyboard or touch and pen actions. For detailed information on specific input methods, refer to our Keyboard shortcuts, gestures, and hotkeys article.

Selecting sketch items and constraints

Selecting a sketch item highlights its connected constraints and dimensions. Constraints highlight all affected elements, making it easier to visualize relationships between sketch elements.

If a sketch is fully defined, all elements remain constrained and will not move unless a constraint or dimension is removed. If the sketch is underdefined, moving an element will adjust its position relative to the constraints applied.

Area Selection

The **Area Selection** tool is triggered by drawing a selection box in the modeling space. The **Area Selection** tool functions differently based on the direction that the selection box is created. There are also filters that can be used to specify what elements you want to include in the selection. For more information, explore the sections below:

- Selection box direction
- Using filters

Selection box direction

You can control what is included in the area selection by modifying the direction you draw a selection box:

- **Left to right**
  Creating a selection box from left to right will only select elements that are completely within the selection box.
- **Right to left**
  Creating a selection box from right to left will select all elements that the selection box touches.

Using filters

While making your selection, you have the option to filter the selection based on item type. Options include:

- **All Items**
- **Bodies Only**
- **Faces Only**
- **Edges Only**

Keyboard and mouse/trackpad

1. Keep the selection box active by holding the mouse/trackpad button.
2. Adjust the filter using these methods:
    - Press the **Tab** key to cycle through the selection filters.
    - Use the following keyboard shortcuts:
        - **B** = Bodies Only
        - **F** = Faces Only
        - **E** = Edges Only

Touch and pen

1. Keep the selection box active by holding down on your touch device.
2. Use your finger to tap the selection filters.

Advanced selection

Use advanced selection options to work more precisely in complex models. These tools help you isolate geometry, access hidden items, and resolve overlapping selections more efficiently.

On-top selection and hover

Selected items remain visible even when overlapped by other geometry. This makes it easier to keep track of selections and navigate complex designs.

Select from overlapping items

When multiple items overlap in the same area, a pop-up list appears so you can select the exact sketch, face, or edge.

To select items more easily:

- Zoom in to better separate overlapping items in the pop-up list.
- Name sketches, faces, and bodies to make them easier to identify. If an item is unnamed, the pop-up will display its default name.
- You can use this method when working with dense or layered geometry for more precise selection.

Select Through mode

Access internal geometry without hiding or adjusting surrounding bodies. Use the context menu or keyboard shortcut to enable Select Through mode and select through visible geometry for faster edits.

Notable points

Notable points, also known as snapping points or object snaps, are points to help you identify specific locations within sketches or bodies. These points are particularly useful when making measurements, using constraints, connecting sketches, or aligning 3D bodies. You can turn on and turn off the visibility of these points from the **Snap To** settings.

Types of notable points

Endpoints

Endpoints are found in linear, arc, and spline-based sketch elements or edges of bodies. Faces of 3D bodies inherit all the notable points from their bounding edges.

Midpoints

A midpoint is the middle point between two endpoints of a line segment. Midpoints can be found on linear sketch elements or edges.

Center points

Center points are points at the center of arc and circular edges, and planar rectangular faces. They can be found on cylindrical and conical faces of 3D bodies.

Intersection points

An intersection point is the point at which two lines meet or intersect. Intersection points can be found on linear sketch elements.

> **Tip**
>
> Splines, no matter how symmetrical, don't have midpoints or center points. If these points are missing an item, check if the element has a measurable radius or diameter to make sure that it is not a spline.

Accessing tools

Shapr3D has an innovative user interface that provides several methods for accessing tools. You can choose the method that best matches your project workflow or preference below.

- Menus
- Project Sidebar
- Adaptive user interface
- Selection-based modes
- Items Manager
- Keyboard shortcuts and hotkeys
- Command Search
- Context menus
- Menu bar

Menus

You can find the menus on the sides of your modeling space, floating over the canvas, keeping your project front and center. The main menu is on the left side of the screen and sketch constraints appear on the right side.

Sketching and modeling tools are organized into submenus, and the main menu provides convenient access to the Project Sidebar and Items Manager.**Note for iPadOS:** If you're left-handed or simply prefer to have the main menu on the right side, you can change the interface settings in **Settings**.We

Project Sidebar

The Project Sidebar is a quick way to navigate between the modeling, Visualization, and 2D Drawings space. Simply select **Modeling**, **Visualization**, or **Drawing** from the main menu to open the Project Sidebar.

Under the **Project Sidebar**, you can find the following options:

- **Modeling** – Opens the modeling space.
- **Visualization** – Opens the Visualization space.
- **Drawings** – Open existing 2D Drawings listed under it, or right-click on the 2D drawings to rename, duplicate, or delete them. To create a new drawing, select +.

Adaptive User Interface

As you make selections in your project, Shapr3D predicts what tools you want to use next and makes them available at the left. Using this adaptive user interface is a fast and efficient way to develop your projects. This method allows you to focus on your model, rather than on browsing through menus.

To use the adaptive UI for accessing tools, make selections in the modeling space or **Items Manager**, then make use of the default tool that becomes active or choose a tool from the customized menu at the left. For more information, see **Adaptive User Interface**.

> **Tip**
>
> To access additional tools from the adaptive menu, select **More**.

Modes

You can find modes in the bottom-left corner of the main menu that you can easily turn on or turn off. If you don't have any selections, **Section View** and **Measurement Mode** are readily available by default. As you make selections or access relevant tools, the modes adapt to provide you the tools you need.

Items Manager

The Items Manager is another quick way to trigger tools whenever you select items. For example, selecting a body from the available items triggers the adaptive UI with available tools you can use. To learn more, visit **Items Manager**.

Keyboard shortcuts and hotkeys

The most frequently used tools can also be selected via keyboard shortcuts. The shortcuts are highlighted next to the name of the tool in the side menu. To learn more about shortcuts, visit **Keyboard shortcuts, gestures, and hotkeys.**

Command Search

The **Command Search** function allows you to access tools by searching for the command name. As you type into the search field, valid commands appear in the list below. You can launch tools directly from the results list.

Depending on your platform and settings, **Search Commands** is available via the 'X' or Control/Command + F keyboard shortcuts, from the toolbar or the help menu.. For more information, see Command Search.

> **Tip**
>
> You can pre-select elements in your design before searching for commands. When you pre-select items, the search results are customized based on your selection.

Context menus

If you're using a mouse or trackpad, using context menus is another quick way to access tools and commands. Simply secondary click (right-click) anywhere in your modeling space to open the pop-up menu containing commands, actions, or options relevant to your workflow.

To learn about the different context menus available, visit Context menus.

Menu bar

In Shapr3D Windows, you can access basic tools and commands from the file menu in the top navigation bar. Simply select **File, Edit, View**, and **Help** to find shortcuts to basic commands.

In Shapr3D Mac, you can also access modeling tools from the Mac menu bar. Simply go to the menu bar and select the options for **Sketch**, **Add**, **Transform**, and **Tools** to access modeling tools, and select **File**, **Edit**, **View**, **Window**, and **Help** find shortcuts to basic commands.

Move and copy items using the gizmo

The gizmo provides directional control for several tools within Shapr3D. The parts of the gizmo are detailed below.

Gizmo center

The gizmo center has two functions:

- The gizmo center defines the center of any rotational movement.
- The gizmo center controls the orientation of the gizmo.

You can move the gizmo center by dragging. The gizmo center will snap to existing geometry such as axes, faces, edges, sketch profiles, and construction geometry, allowing you to define a new center for any rotational movement or reorient the gizmo.

> **Tip**
>
> Some tools include an **Auto-orientation** option that can be turned on and off from the upper left of the modeling space. Turning off this option prevents the axes of the gizmo from aligning with other geometry.

Arrows

Gizmo arrows provide linear and rotational controls. Arrows can be used in two ways:

- Drag an arrow to move in a direction dynamically.
- Select an arrow to enter a precise value for movement in the selected direction.
  Note: To learn more, visit defining and calculating numerical values.

History settings

In the History sidebar, your linear and rotational controls are defined as a **Movement/Rotation** step.

Tiles

Gizmo tiles can be dragged for linear movements along a plane.

Dimension label

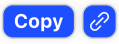
Dimension labels appear next to a selected gizmo arrow and they display the current value for the movement. To modify the value of a dimension label:

- **Keyboard and mouse/trackpad**: Hover your pointer over the arrow and click the dimension label to enter a value. To use the numpad, click the dimension label to select it and then click again to activate the numpad.
- **Touch and pen**: Tap the dimension label to modify the value using the numpad.

Copy badge

The **Copy** badge  can be used to to create a copy or multiple copies of a selected item.

For tools that use the **Copy** badge, the color of the badge indicates when it is selected:

- Unselected: 
- Selected: 

To use the **Copy** badge:

1. Select the item to copy.
2. Select the **Copy** badge  in the gizmo.
3. Drag the linear or rotational arrows to where you want the copied item to be.
4. To create multiple copies, keep the **Copy** badge turned on and continue moving the arrows to the places where you want the copies to be.
5. **Optional:** To unlink you copy action from History, turn off the **Link** badge  .
6. To finish, turn off the **Copy** badge or select an empty area in the grid.

History settings

In the History sidebar, items you copy are defined as a **Movement/Rotation** step.

You can modify the Movement/Rotation step parameter to turn on or off your copy action.

Note: For unlinked copies, History displays a step called **Unlinked Copy**.

Defining and calculating numerical values

When modeling, you can quickly define or calculate numerical values such as dimension constraints using the expression input field or the built-in numpad/calculator. These fields support basic arithmetic operations, allowing you to perform simple calculations directly in Shapr3D

For more advanced calculations, including unit conversions and order of operations, visit Variables and expressions.

Entering numerical values

You can define values in both 2D and 3D views by entering them directly into parameter labels using the parameter input field or numpad/calculator.

Note: Avoid commas as decimal separators to prevent errors.

To define numerical values while modeling:

Keyboard and mouse/trackpad

1. Hover over the tool parameter label and enter a value to automatically open the input field.
2. To finalize, press the **Enter** key.

Touch and pen

1. Tap the tool parameter label to open the input field.
2. Tap the numpad button to open the numpad/calculator.
3. Tap the numbers to enter a value.
4. To finalize, select the checkmark.

Perform basic arithmetic

You can enter simple calculations directly in the field or use the built-in numpad/calculator when defining numerical values.

Input field

To perform a basic calculation:

1. Enter an arithmetic expression directly in the field.
2. Press **Enter**, and the evaluated result will replace the expression.

Example: Entering 10 + 5 automatically displays 15.

Supported operations:

- Addition (+)
- Subtraction (-)
- Multiplication
- Division (/)

Numpad/Calculator

To perform calculations using the numpad/calculator:

1. Enter values and calculations as you would on a standard calculator.
2. To finalize the result, select the **checkmark**.

Read more

- To learn more about dimension labels, visit Editing sketch dimensions.
- For more advanced calculations, including unit conversions and order of operations, visit Variables and expressions.

Boolean operations

Boolean operations in CAD, commonly known as the **Union**, **Subtract**, and **Intersect** commands, are a set of 3D modeling tools that can merge, separate, or create geometry from existing bodies. These tools allow you to create complex shapes and forms of your models.

In Shapr3D, boolean tools have a **Keep Originals** option that you can use to keep the original bodies of your model after completing the operation. Keeping your original bodies can be useful for later use or if you want to keep a backup for each part.

Another tool that uses boolean operations is **Extrude**. When creating an extrusion, the **Extrude** tool provides you with a boolean badge with the options for **Union**, **New Body**, **Subtract**, and **Intersect** to help you easily choose your preferred extrude type.

Read more

To learn more about each boolean tool and the different settings for their **Keep Originals** option, visit Union, Subtract, and Intersect**.**

To learn more about other available 3D modeling tools, visit the Tools menu**.**

Sketch controls

Sketch states, patterns, and dimensioning

Sketch states

Sketch states in Shapr3D provide a visual representation of the definition level of sketch items and points. These visual cues help determine whether an item is fully constrained or still has available degrees of freedom, ensuring better control over sketch behavior.

Degrees of freedom

A sketch item has **degrees of freedom** when it can still move in at least one direction. In Shapr3D, sketch items have three available degrees of freedom:

- **Rotation:** Can turn clockwise or counterclockwise.
- **Horizontal movement:** Can move left or right.
- **Vertical movement:** Can move up or down.

A sketch is fully-defined when all degrees of freedom are fixed by constraints or dimensions. If an item remains under-defined, it retains some movement until further constraints are applied.

Sketch states

Fully-defined sketches (Green)

A sketch is **fully-defined** when:

- All its items have constraints or dimensions applied.
- It has no remaining degrees of freedom.
- Its position, size, and relationships are completely determined.

A fully-defined sketch remains fixed unless a constraint or dimension is modified or removed.

Under-defined sketches (Blue)

A sketch is **under-defined** when:

- It lacks constraints or dimensions.
- It has at least one degree of freedom remaining.
- It can still move, rotate, or resize.

Under-defined sketches require additional constraints or dimensions to become fully-defined.

> ### Tip
>
> To quickly check the state of a sketch, look at the color of the sketch items. Green means fully-defined, while blue means under-defined.

Sketch point states

Fully-defined points (Green)

A sketch point is fully-defined, fixed, or locked when:

- It has no remaining degrees of freedom.
- It stays in place and does not move when other sketch items are adjusted.
- It acts as an anchor for connected sketch items.

Under-defined points (Blue)

A sketch point is under-defined, movable or unconnected when:

- It still has at least one degree of freedom.
- It can be repositioned freely.
- It is not constrained to another sketch item.

Connected points (Shared color with sketch item)

A connected point follows the state of the item with the greater degree of freedom:

- If both connected items are fully-defined, the shared point is also fully-defined.
- If one connected item is under-defined, the shared point remains under-defined until constraints or dimensions are applied.

Types of connected points:

- **Coincident constraint:** The point is connected to a line, curve, or their extended path , allowing it to move along the element, even beyond its visible endpoints.
- **Midpoint constraint:** The point remains at the center of a line even if the line's length or position changes.
- **Point-to-point connection:** The point is connected to another sketch point, and both move together.

Understanding degrees of freedom in connected sketches

If a fully-defined item is connected to an under-defined item:

- The fully-defined item remains fixed.
- The under-defined item can still move within its remaining degrees of freedom.
- The shared point between the two behaves based on the item with the highest degrees of freedom.

Read more

Learn more about defining sketch relationships in Constraints overview and adjusting dimensions in Editing sketch dimensions.

Sketch pattern constraint

When a **Pattern** is created within a sketch or a sketch profile is patterned in the 3D environment, a sketch pattern constraint is automatically created. If your constraint visibility setting is turned on, the source sketch from which you created your pattern displays a sketch pattern constraint badge.

A sketch pattern constraint maintains the relationship between the patterned elements so that changes you make to one element also apply to the other elements in the pattern.

Modify pattern definition and quantity

You can quickly modify the pattern definition, distance, and quantity of patterned elements as long as there is an existing constraint.

To modify pattern definition and quantity, follow these steps:

1. Select any sketch belonging to a sketch pattern to re-activate the pattern control badges.
2. Use the pattern control badges to adjust the pattern definition type, distance, and quantity.
3. To finish, select an empty area of the grid or select **Exit Sketching**.

> ### Tip
>
> The sketch pattern constraint can only be created using the **Pattern** tool for sketch elements or sketch profiles. If a sketch pattern constraint is deleted, it will permanently break the relationship between pattern instances.

Delete sketch pattern constraint

If you want to modify elements in your sketch pattern individually, you can break the pattern by deleting the sketch pattern constraint.

To delete a sketch pattern constraint, follow these steps:

1. While in sketch mode, locate the source sketch from which you created your pattern.
   Note: If your constraint visibility setting is turned off, select or hover over the source sketch
2. Select the sketch pattern constraint badge on the source sketch.
3. In the **Sketch** menu, select **Delete Constraints**.
4. Modify individual elements from the pattern.

Video walkthrough

Shapr3D Manual - Using pattern constraints | Sketching

> ### Read more
>
> To learn more about the Sketch Pattern tool, visit Pattern (Sketch).

Editing sketch dimensions

Each sketch element has a dimension label where you can view and edit the dimensions and measurements of sketches.

Edit a dimension

Keyboard and mouse/trackpad

1. Click the element to select it.
2. Hover your pointer over the dimension label.
3. Use your keyboard to type the value of the new dimension or click the dimension label again to open the numpad.
4. **Optional:** You can enter simple mathematical expressions (Example: `12+34` or `50/2`) into the dimension label or use operators of the numpad to calculate the final value of the dimension.

Touch and pen

1. Tap the element to select it.
2. Select the dimension label.
3. Use the numpad to input the new dimension, or use the calculator to calculate the final value of the dimension.
4. To apply the value, tap the check mark.

> **Read more**
>
> To learn more about modifying numerical values, visit Defining and calculating numerical values.

Dimension types

The types of sketch dimensions are length, diameter, radius, and angle.

Length

The length is the distance between two endpoints such as a line.

You can define the length of two endpoints with the following distance types:

- **Absolute** – Distance between the two endpoints of a line.
- **Horizontal** – Distance between the two line endpoints relative to the horizontal axis.
- **Vertical** – Distance between two line endpoints relative to the vertical axis.

To define a distance type, select the **Distance Type** badge that appears beside the dimension label.

Diameter

The diameter is the distance between two points in a circle that passes through the center.

You can define the diameter of circles.

To define a circle by its diameter, your **Circular Annotation** in Settings or Preferences must be set to **Radius and Diameter**.

Radius

The radius is the distance between the center of a circle to any endpoint. For circles, the radius is half of the diameter. Meanwhile, ellipses have two radii: major and minor, where the major is the larger of the two.

You can define the radius of circles, arcs, and ellipses.

To define a circle by its radius, your **Circular Annotation** in Settings or Preferences must be set to **Always Radius**.

Angle

The angle is formed by the common point of two intersecting lines.

You can define the angle of the following sketch elements:

- Between two connected or unconnected lines
- Between two connected splines or arcs
- Between a connected line and arc or spline

To change the angle between two elements, select them and then enter the new value of the angle.

Video walkthrough

Editing sketch dimensions | Sketching

Using sketch planes

To sketch in 2D, you must define a sketch plane where you want to create your sketch. You can define a sketch plane in the following ways:

- By selecting a plane before you start sketching
- By selecting a planar face or construction plane
- From the **Orientation Cube**
- From **Views** in **Views and Appearance**

How to start a sketch in different planes

Keyboard and mouse/trackpad

To start sketching on a plane:

1. From the main menu, select **Sketch**.
2. From the three small rectangles that appear near the origin, click on the rectangle corresponding to the plane you'd like to sketch on.

To select a sketch tool right away:

1. From the sketch menu, select a sketch tool.
   For other methods of accessing tools, visit Accessing tools.
2. From the three small rectangles that appear near the origin, drag your pointer to hover over the rectangle corresponding to the plane you'd like to sketch.
3. Press the **Space bar**.

To start sketching on a different plane:

1. Drag your pointer to hover over a planar face or construction plane.
   Note: This is also possible with your pen if the platform supports hovering with the pen.
2. Press the **Space bar**.

> ### Tip
>
> If you accidentally rotate your view while in sketch mode and want to continue sketching in 2D with a normal view on your previously selected plane, select **Normal to Sketch** at the top.

Touch and pen

To start sketching on a world plane:

1. From the main menu, select **Sketch**.
2. From the three small rectangles that appear near the origin, tap on the rectangle corresponding to the world plane you'd like to sketch on.
   Note: You can also double-tap on the rectangle with your finger.

To start sketching on the grid or an existing plane:

- Using a finger, double-tap on the grid, a planar surface or construction plane.
-

> ### Tip
>
> If you accidentally rotate your view while in sketch mode and want to continue sketching in 2D with a normal view on your previously selected plane, select **Normal to Sketch** at the top.

How Shapr3D organizes your sketches

Sketches you create are added to the Items Manager and the History sidebar as new items. If you continue sketching on a plane immediately after creating a sketch on the same plane, Shapr3D will edit the previous sketch. In every other case, a new sketch is created.

Note: For Shapr3D versions earlier than 5.590, sketches are organized by planes, so adding a new sketch item to a plane you previously sketched on automatically adds it to the existing **Sketch plane** item in the Items Manager.

Video walkthrough

Shapr3D Manual - Defining sketch planes | Sketching

Changing a sketch plane

While in sketching mode, you can move your sketch along the planes using **Move/Rotate**.

To move your sketch along the planes:

1. Double-click/tap the **Orientation Cube** to set the view to default.
   Note: To learn about the different ways to change the view, visit **Orientation Cube** or **Views and Appearance**.
2. Select **Move/Rotate**.
3. Drag the tiles in the gizmo center to move your sketch along the planes.

Read more

To learn more about the **Move/Rotate** tool, visit Move/Rotate (Sketch) and Move/Rotate (3D).

Project spaces

Project environments
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Modeling

[Article link | Table of contents | Back | Next]

The modeling space  , also referred to as project space, design space, or workspace, is your main working area when creating 2D sketches and 3D models. Whenever you start a new project, you are automatically brought to the modeling space. From this space, you can navigate between the Visualization space and 2D Drawings space through the Project Sidebar.

Read more

To learn more, visit Shapr3D modeling space.

Visualization

Visualization  lets you create custom, real-time renders of your models. With over 100 materials, several environment options, and a customizable camera and depth of field settings, you can give your 3D model a realistic look and finish that allows you to share stunning projects with anyone.

Access the Visualization space

To use **Visualization**:

1. Make sure that you have a 3D model in your modeling space.
2. From the main menu, select **Modeling** to open the **Project Sidebar**.
   *You can also access this tool by selecting **Tools** > **Visualize,** or by making selections and using the adaptive menu. For more information and additional methods, visit **Accessing tools**.*
3. From the Project Sidebar, select **Visualization**.
4. Select  to show the properties, and choose between the **Material**, **Environment**, and **Camera** tabs to start modifying the visualization of your model.

> **Tip**
>
> If you use Visualization with an empty modeling space, you can find an option to import a project or a sample model.

Material

You can assign and change the material of your model from the **Material** tab. Depending on the material, you can also adjust the color, transparency settings, and material scaling.

Add a material

To add a material to your whole model:

1. Scroll through the **Material** tab to choose a material to add, or use the drop-down for **All Materials** to choose a category.
2. Drag and drop your chosen material on the whole body of your model.
3. Select an empty space to finish or select **Modeling** from the menu to go back to the modeling space.

To add material to parts of your model:

1. Select a face or body to open **Properties**.
2. Under **Material**, select **Change**.
3. Drag and drop your chosen material to your selected face or body.
   Note: You can also click or tap on the material to automatically apply it to your selected face or body.
4. Select an empty space to finish or select **Modeling** from the menu to go back to the modeling space.

At the top part of the **Material** tab, you can find the **Used Materials** section to view the materials you previously chose for your model.

Change a material

You can change the material of your body from **Properties** in the **Material** tab.

To change a material:

1. In the **Used Materials** section, select the material you want to change (iPadOS/macOS) or click **Edit** (Windows) beside the material to open **Properties**.
2. Under **Material,** select **Change.**
3. Scroll to choose a new material to add, or use the drop-down for **All Materials** to find your desired material.
4. Drag and drop your chosen material on your model.
   Note: You can also click or tap on the material to automatically apply it to your model.
5. Select an empty space to finish or select **Modeling** from the menu to go back to the modeling space.

To reset a material to default:

- **Windows/macOS**: Secondary click (right-click) on a material in the **Used Materials** section, then select **Reset to default material**.
- **Touch and pen**: Swipe the material to the left in the **Used Materials** section.

Search for a material

Quickly find and apply the appearance you need by searching for materials directly in the **Material** tab.

- Use the search bar at the top to type in a material name or finish (e.g. “Aluminum,” “Oak,” “Matte”).
- As you type, matching materials will be filtered instantly.
- Select a material to apply it to your selected body or face.

Adjust material properties

You can adjust the visual characteristics of applied materials to achieve specific aesthetic or technical requirements from **Properties** in the **Material** tab.

To access **Properties,** select a material (iPadOS/macOS) or select **Edit** (Windows) beside the material under **Used Materials**. Learn about the different fields available in **Properties** below.

Note: Depending on the material, some properties may not be applicable.

- **Transmission** – Drag the slider to adjust how much light passes through the material. Higher values create a more glass-like effect.
- **IOR (Index of Refraction)** – Adjust how much light bends as it passes through the material. Higher values simulate denser materials like glass or plastic.
- **Scale** – Drag the slider to change the scale of your material.
- **Roughness** – Drag the slider to adjust the roughness of your material.
- **Auto Match to Body** – Turn on to find the best material mapping based on the orientation of the body. Turn off to fix material mapping to the XYZ axes.
- **Rotate X** – Rotate the material texture about the relative X (main) axes of all the bodies that this material is assigned to.
- **Rotate Y** – Rotate the material texture about the relative Y (secondary) axes of all the bodies that this material is assigned to.
- **Rotate Z** – Rotate the material texture about the relative Z (normal) axes of all the bodies that this material is assigned to.
- **Color Field** – Adjust the brightness and saturation level of the selected color.
- **Hue Slider** – Pick the color you want to apply to faces and bodies.
- **Opacity** – If applicable, move the handle down on the slider to change the opacity or transparency of a body or face. This setting allows you to show the interior of bodies.
  Note: To make a material transparent in the modeling space, apply the **Default Material** or **Simple Transparent Material** from the **Utility** material category.
- **Color** – Available for materials that support color customization. Use this section to define or adjust the color of a material.
  - **Hexadecimal** – Enter a six-digit hex code (for example, #F2F2F2) to apply an exact color.
  - **Pick Face Color** – Samples the color from any face in your model and applies it to the selected material.
  - **Color Swatches** – Store and reuse colors you often apply. Choose or sample a color, then select an empty swatch to save it. To delete a saved color, secondary click (right-click) the swatch and select **Delete**.
  - **Default Body Color** – Sets the color applied to all new bodies that don’t have a custom color or material assigned.

Custom materials

Shapr3D Visualization supports importing custom materials, making it easy to bring an existing material library from other tools into your projects.

Custom materials are imported using the **GLB** file format, which is commonly supported by third-party material and rendering applications such as Blender or KeyShot. For a more basic custom material using a single texture, you can also import **JPEG**, **JPG**, or **PNG** image files. Once imported, the materials become available in Visualization for realistic rendering and presentation.

When importing a custom material file:

- Only the material definitions are imported. Any geometry included in the file is ignored.
- If the file contains multiple materials, all of them are imported and added to the **Material** tab.
- In addition to **GLB**, you can also import **JPEG**, **JPG**, or **PNG** files to create a basic custom material using a single texture.
- Imported materials are saved within the current project and sync across all devices, but aren't automatically available in other projects. To reuse them elsewhere, import the material file again.

To import a custom material:

1. Create or manage your materials in an external tool (for example, Blender or KeyShot).
2. Open the **Material** tab.
3. Scroll to the **Custom** section.
4. Select **Import...** and then choose the **.GLB** file to add.

Once the custom material is imported, you can apply and adjust the material settings like how you would with any other material.

Decals

Shapr3D Visualization supports importing decals, making it easy to apply custom graphics, logos, or labels directly onto your model surfaces for realistic rendering and presentation.

Decals are imported as image files. Once imported, they become available in Visualization and can be placed directly onto any surface by dragging them onto the model.

Import and apply decals

1. Open the **Material** tab.
2. Scroll to the **Decals** section.
3. Select **Import...** and then choose the image file to add.
4. Once imported, drag the decal onto the surface of your model to place it.

Adjust decal settings

To modify a decal, select it to open the properties panel, then click or tap **Edit** next to the specific decal. Available properties depend on the decal type and are detailed below:

- **Scale** – Adjusts the physical size of the decal on the body while maintaining its proportions.
- **Rotate** – Rotates the decal clockwise or counterclockwise on the selected face.
- **Opacity** – Controls the transparency of the decal, ranging from completely solid to fully transparent.
- **Wrap adjacent faces** – Extends the decal over corners and edges onto connected neighboring faces.

- **Cylindrical projection** – Wraps the decal evenly around curved or cylindrical bodies to prevent stretching and distortion.

Environment

You can change the environment of your model from the **Environment** tab.

Select an environment

To add an environment, go to the **Environment** tab and select an environment.

Adjust environment settings

Fine-tune the lighting and background of your workspace to better present your model. To adjust your environment settings:

1. Select the settings icon at the bottom-right corner of the current environment.
2. From **Properties,** you can adjust the following:
    - **Light Rotation** – Drag the slider to change the orientation of shadows.
    - **Sun Elevation** – Drag the slider to adjust the vertical angle of the light source above the horizon. A lower value simulates early morning or late afternoon lighting, while a higher value creates overhead sunlight.
    - **Light Intensity** – Drag the slider to adjust the light intensity of your environment.
    - **Ground Plane** – This adds an infinite plane to the scene that receives shadows from your models. When enabled, you can choose between **Snap to Model** or **XY Plane.**
3. Select **Done** to finish.

Camera and Depth of Field

You can select a surface point on a body to add a realistic depth of field effect to your visualizations.

Add Depth of Field

When you apply a depth of field effect, parts of the screen at a similar distance from the camera will remain sharp, while areas closer or further away will become increasingly blurred.

To add a depth of field effect to your model:

- **Keyboard and mouse/trackpad:** Secondary click (right-click) on a part of a body to open the context menu. Use **Set as Focus Point** to bring it into focus. Use the visual indicator that pops up to help you bring a part into focus. Use **Remove Focus Point** to turn off **Depth of Field**.
- **Touch and pen:** Tap on a part of a body to bring it into focus. Use the visual indicator that pops up to help you bring a part into focus. To turn off **Depth of Field**, select a focus point anywhere outside a body.

Adjust Camera and Depth of Field properties

You can adjust the following **Depth of Field** properties in the **Camera** tab:

- **Field of View** – Adjust the field of view angle of the camera.
- **Aperture** – Define the depth range where the image remains sharp in the depth of field effect.
- **Blur Intensity** – Adjust the overall intensity of the depth of field effect.

Sharing your projects

- **Capture your projects using the Capture tool**
  - You can capture screenshots in Visualization to share with anyone. To learn how, visit Capture your projects via Visualization.
- **Export your projects**
  - **iPadOS and macOS users**: you can export your workspace contents to a textured USDZ file via the export menu. To learn how, visit Export.
- **Augmented Reality**
  - **iPadOS users**: If you want to see your model represented in a physical environment via augmented reality, select **AR** at the top left corner of the screen to export your model in USDZ format. For more details on using augmented reality with Shapr3D, visit Augmented Reality.

Export custom materials for Shapr3D Visualization

Shapr3D Visualization supports importing custom materials through the **GLB** file format, as well as **JPEG**, **JPG**, and **PNG** image files for creating more basic custom materials using a single texture. This makes it possible to bring in materials created in external rendering or material-authoring applications and use them directly in Shapr3D.

A **GLB** file is based on the glTF (GL Transmission Format) standard and is designed for exchanging 3D assets between applications. It can bundle material definitions, textures, and shading information into a single file, which makes it a practical format for transferring custom materials into Shapr3D.

Many professional visualization tools support exporting materials as GLB, allowing you to build custom material libraries outside of Shapr3D and then import them for realistic rendering and presentation.

This article outlines common workflows for exporting materials from popular rendering tools and preparing them for import into Shapr3D Visualization.

General workflow

This section outlines the general workflow for preparing and exporting a .**GLB** file from a third-party tool, ready to import into Shapr3D Visualization.

Most rendering and material-authoring tools don't export materials as standalone files. Instead, materials are included in a **.GLB** export by applying them to one or more bodies first.

1. Create or edit your material in a supported third-party application such as Blender, Substance, or KeyShot.
2. Apply the material to a body in the scene. If you want to export multiple materials, apply each one to a separate body.
3. Export the textured body or bodies as a **.GLB** file. The exported GLB file will carry the material definitions and textures.
4. Import the GLB file into Shapr3D Visualization.

For detailed platform-specific export steps, refer to the official documentation of your rendering software.

> **Tip**
>
> If your rendering tool doesn't support direct GLB export, you can often use Blender as a conversion step by exporting an intermediate file format (such as FBX) and then exporting from Blender as GLB.

Workflow by application

Blender

Blender is one of the most common tools for preparing GLB-based material libraries. You can create materials directly in Blender or use it to convert materials from other software into the GLB format.

- Export using **File > Export > glTF 2.0 (.glb)**
- Learn more from Blender's documentation: glTF 2.0 exporter guide

Substance (Adobe)

Substance tools can generate advanced PBR material sets that can be exported and packaged into GLB through compatible workflows.

- Export textures and materials, then assemble into GLB using a supported pipeline such as Blender or another renderer.
- Refer to Adobe's official documentation:
  - Substance 3D documentation
  - First Steps with Substance 3D Sampler

KeyShot

KeyShot materials can be transferred into GLB workflows depending on your export setup and supported extensions.

- Export your material setup into a compatible interchange format, then convert to GLB if needed.
- Refer to KeyShot support resources:
  - How to use smart export options in KeyShot
  - Export formats

Autodesk VRED

VRED workflows depend on the version you are using:

VRED 2024 and newer

1. Apply the materials you want to import to a body or multiple bodies within your VRED scene.
   Note: To export multiple materials, apply each one to a separate body.
2. Export the material as a **GLB** file from VRED.
3. Import the GLB file into Shapr3D Visualization.

VRED versions before 2024

1. Export an **FBX** file from VRED.
2. Import the FBX file into Blender.
3. Export from Blender as **.GLB**.
4. Import the GLB file into Shapr3D Visualization.

For more details, refer to Autodesk's documentation: Best Practices for Sharing and Embedding USDz and GLB Files.

Generate a GLB material file using AI tools

If you already have an image reference (such as a wall texture or fabric pattern), you can also use AI-based tools to help generate a **.GLB** file that contains a ready-to-import material. This workflow can be useful for quickly creating custom materials when you don't have an existing rendering pipeline.

1. Upload your reference image to an AI tool such as ChatGPT that supports 3D or material export workflows.
2. Use a prompt requesting a GLB file that includes a material with embedded textures.
    - For example: *Generate a .glb file that contains a material with 2K textures using this image as a reference.*
3. Download the generated GLB file.
4. Import the GLB file into Shapr3D Visualization.

Use external material libraries

If you don't already have a material library, you can also find free material libraries online from sources such as Poly Haven. If a material isn't available in .GLB format you can:

1. Download a Blender-compatible version.
2. Open the file in Blender.
3. Export the file as a .GLB file.
4. Import the GLB file into Shapr3D Visualization.

> **Read more**
>
> To learn how to import and apply custom materials, visit Visualization.

2D Drawings

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The **2D Drawings** is where you create detailed technical drawings from your 3D models. These drawings provide precise dimensions, annotations, and views to communicate design intent for manufacturing, engineering, or documentation purposes. All drawings are saved in the Project Sidebar, where you can access, organize, and manage them efficiently.

Add a 2D Drawing

To add a new 2D Drawing:

1. From the **Project Sidebar**, select + beside **Drawings**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the 3D body for which you want to create a 2D drawing.
3. Select **Next**.
4. In the **Drawing Preferences** dialog, specify the following settings:
   - **Drawing Title** – Enter a name for your 2D Drawing.
   - **Orientation** – Choose between portrait and landscape.
   - **Sheet Size** – Choose from ISO or ANSI standards.
   - **View-to-Sheet Scale** – Choose your preferred scale.
   - **Include 4 Views** – Enable to automatically add the front, left, top, and isometric views of your model in the drawing sheet.
     Note: The front view is the base view by default, while the other views are the projection views.
5. Select **Continue** to generate the drawing sheet.
6. Optional: Adjust the following details according to your preference:
   - Scale of a drawing – Double-click/tap on a drawing and select the scale dimension label 1:2 to choose a different scale.
   - Show hidden lines – Double-click/tap on a drawing and select ⬚.
   - Title block information – Double-click any text in the title block to change it to your preferred text.
   - Drawing Properties – Select ▭ or ▭→ to update your sheet settings.
     - **Sheet**
       - **Orientation** – Choose between portrait and landscape.
       - **Sheet Size** – Choose from ISO or ANSI standards.
       - **View Scale** – Choose for preferred view-to-sheet scale.
       - **Projection** – Choose your preferred projection angle.
       - **Title Block** – Choose your preferred layout. To learn more, see the Title block layouts section below.
     - **Dimensions**
       - **Units** – Choose your preferred unit.
       - **Angle Format** – Choose your preferred angle format.
       - **Length Precision** – Choose your length precision format.
       - **Angle Precision** – Choose an angle precision format.
       - **Decimal Separator** – Choose between a comma and period for your preferred decimal separator.
     - **Line Widths** – Choose your preferred line width for any of the following:
       - **Visible Outlines**
       - **Hidden Lines**

- **Dimension Lines**
- **Center Lines**
- **Section Lines**
- **Detail Marks**

Video walkthrough

Shapr3D Manual - Create new drawings | 2D Drawings

Open or manage existing 2D Drawings

All 2D Drawings are saved in the **Project Sidebar**, where you can access and manage them at any time.

To open or manage an existing 2D drawing:

1. Open the Project Sidebar.
2. Under **Drawings**, find the 2D drawing to open.
3. Select the 2D drawing to automatically open it, or right-click on the drawing to rename, duplicate, or delete it.

Title block layouts

A title block is a table that you can usually find at the bottom-right corner of 2D Drawing sheets. The main purpose of a title block is to provide important information that can help readers understand your 2D Drawing better.

In Shapr3D, you can choose from a variety of predefined title block layouts to personalize your drawing sheet. You can also update the fields in your title block with any information you need.

By default, title block layouts contain the following information:

- Title
- Units
- Scale
- Projection Angle
- Size
- Last Updated Date
- Sheet Size

To choose a layout, go to **Properties** > **Title Block**, then choose any of the following:

- **Simple** – This is the default layout when creating a 2D Drawing. The general information is displayed in the bottom-right of the sheet.
- **Empty Sheet** – This layout completely removes the title block and border.
- **Border Only** – This layout removes the title block and only displays the default border.
- **Horizontal** – The general information is displayed horizontally at the bottom of your sheet.
- **Vertical** – Displays your title block vertically on the right side of your sheet. This layout provides an additional table where you can record **Revision**, **Name**, and **Date** information.
- **Block** – Displays your title block in the bottom-right of your sheet. Similar to the vertical layout, this layout provides an additional table where you can record **Revision**, **Name**, and **Date** information.

- **Block with Table** – Displays the block layout with an additional table to the left that you can customize with information you want such as details about model parts or important features that you want to highlight.

Video walkthrough

Shapr3D Manual - Title block layouts | 2D Drawings

Views

You can add multiple projections in 2D Drawings to highlight every detail of your model. These projections are created from any of the following view types:

- Base view
- Projection view
- Section view
- Detail view

Base View

The base view is the main building block of your 2D drawing. Additional views you create always reference back to your base view.

In Shapr3D, projected 2D views align automatically with their base view. When you move a 2D base view, all its 2D projected views also move in alignment. The same applies to 2D projected views: when you move a 2D projected view, its 2D base view also move in alignment.

You can add a new base view with any of the following orientation: front, top, right, back, left, bottom, isometric.

2D base views

To add a 2D base view of your model:

1. From the 2D Drawings menu, select **Views** > **Front**.
2. **Optional**: From the Base View popup, select **Custom base view…** to change the projection angle from which your drawing is created
3. Choose a 2D orientation for your base view.
4. **Optional**: To update your reference bodies, select **Bodies…** > Select your new reference bodies > **Done**.
5. Select + to add your base view.
6. From the projection previews that pop up, select + on a projection view you want to add.
7. To finish, select **Done**.
8. To move your base views and projections, double-click/tap on any of them, then use the arrows to drag them to your desired position.

Isometric views

To add an isometric view of your model:

1. From the 2D Drawings menu, select **Views** > **Front**.
2. **Optional**: From the Base View popup, select **Custom base view…**to change the projection angle from which your drawing is created
3. ,Choose an isometric orientation for your base view.
4. **Optional**: To update your reference bodies, select **Bodies…** > Select your new reference bodies > **Done**.
5. Select + to add your isometric view.
6. Use the arrows to drag your isometric view to your desired position.
7. To finish, select an empty space, or select **Deselect All** from the menu.

Add a new projection

To add more projections to your drawings:

1. From the 2D Drawings menu, select **Views** > **Projection**.
2. Select a view from which you want to create a projection.
   Note: If you only have one view, the projection previews automatically pop up.
3. From the projection previews that pops up, select + on a projection view you want to add.
4. To finish, select **Done**.

Section views

You can create section views from base or projection views to visualize the interior of the model.

To create section views of your model:

1. From the 2D Drawings menu, select **Views**.
2. Select **Section View**.
3. Use the reference points as guides to precisely place your section line. This section line defines the plane that cuts through the object.
   Note: If you'd like to replace the section line with a different one, simply draw a new line. The previous section line will be removed automatically.
4. Drag one of the arrows next to the section line to where you want to place your section view.
5. Select an empty space to finish, or select **Close** from the menu.

Detail View

You can add detail views to your 2D Drawings to highlight and annotate your dimensions in a magnified view.

To create detail views of your 2D Drawings:

1. From the 2D Drawings menu, select **Views**.
2. Select **Detail View**.
3. Draw a circle around the area you want to magnify.
   - **Keyboard and mouse/trackpad**: Click on a guide point, then move or drag your pointer to form the circle.
   - **Touch and pen:** Tap on a guide point then drag your pointer to form the circle.
4. Click or lift your pen to finalize the circle.
   Note: Make sure to include the full geometry of the drawings you want to highlight with dimensions.
5. Select **Next** to automatically generate the detail view.
6. Click or tap to place your detail view anywhere in the drawing sheet.
7. Use the arrows or center point to adjust the placement of your detail view.
8. **Optional**: Select 1:2 or  to choose a different scale or show/hide hidden lines for your detail view.
9. Dimension or annotate your detail view as how you would in a regular 2D drawing. Refer to Dimensions or Geometries sections below for more information.

Video walkthrough

Shapr3D Manual - Add views to drawings | 2D Drawings

Dimensions

You can select from different dimension types to add to your 2D Drawings such as line length, point-to-point distance, and arc angle.

To add dimensions to your 2D Drawing:

1. Select **Dimensions** in the drawings menu.
   You can directly select an item in your drawing and the adaptive menu automatically dimensions the item. Select the check mark to confirm the dimension. If there is more than one available dimension, select the dimension before selecting the check mark.
2. Select a dimensioning tool.
   - **Line Length** – Measures the length of a straight line.
   - **Point-to-Point Distance** – Measures the distance between two selected points. These points can be endpoints, intersections, quadrant points, and center marks.
   - **Point-to-Line Distance** – Measures the distance between any point and a line. These points can be endpoints, intersections, quadrant points, and center marks.
   - **Line-to-Line Distance** – Measures the distance between two parallel lines.
   - **Arc Angle** – Measures the included angle of an arc.
   - **3-Point Angle** – Measures the inner or outer angle between two virtual lines defined by one joint vertex and two other points on the legs. These points can be endpoints, intersections, quadrant points, or center marks.
   - **Line-to-Line Angle** – Measures the angle between two non-parallel lines.
   - **Radius** – Measures the distance between an arc or circle's circumference and center.
   - **Diameter** – Measures the length of a line that would cross a circle or arc's center and two points on the shape's circumference.
   - **Min-Max Distance** – Measures the minimum or maximum distance between a circle or arc and another circle, arc, or line. Curves that are not part of a section of a virtual circle or arc are not included.
3. Select the items that need to be dimensioned.
4. Drag the dimension on the sheet to reposition it anywhere you want.
   Note: Dimensions you create snap at equal increments, allowing you to easily match your new dimensions to existing ones, or place them symmetrically at a new location.
5. After you finish adding dimensions, select **Done**.
6. Tap the dimension editor badge that appears beside the dimension to edit any of the following:
   - **Prefix Text** – Enable to enter a prefix text for your dimension. For easy access, you can select any of the commonly used symbols available in the Prefix text field below.
   - **Tolerances** – Enable to enter a size tolerance applicable to your dimension. To choose a tolerance type, select the drop-down menu and choose between Symmetrical, Deviation, Limits, and Basic.
   - **Suffix Text** – Enable to enter a suffix text for your dimension. For easy access, you can select any of the available commonly used symbols below the Suffix text field.
7. Optional: To remove a dimension, select it > **Delete Selection**.

Note: To specify the precise decimal measurement of linear and angular dimensions, select or to open **Properties** > **Dimensions** and then choose **Length Precision** or **Angle Precision**.

Video walkthrough

Shapr3D Manual - Add dimensions to drawings | 2D Drawings

Geometries

You can annotate the non-isometric views in 2D Drawings with geometries using the following geometry types:

- Centerline
- Center mark
- Intersection mark

Centerline

Centerlines are dashed lines that run between two reference points or lines. You can use them in 2D Drawings to:

- Show the axes of circular or cylindrical features such as holes and discs
- Dimension circular features
- Point out features that share the same central axis

Note: Centerlines can't be dimensioned with the **Line length** tool.

To add a centerline, follow these steps:

1. From the **2D Drawings** menu, select **Geometries**.
2. Select one of the centerline geometry tools:
    - **2-Point Centerline** – Select two points as your reference in the drawing.
    - **2-Line Centerline** – Select two lines as your reference in the drawing.
    - **3-Point Circular Centerline** – Select two points as your reference in the drawing.
    - **3-Point Centerline** – Select three points on a circumference.
3. To adjust the centerline, select it and then drag the arrows or points to extend the centerlines created between the reference points or lines.
4. To finish, select **Done**.

Center mark

Center marks are cross-shaped annotations that indicate the centers of circles, arcs, and circular edges. You can also mark these centers as reference points for dimensioning.

To add a center mark:

1. From the **2D Drawing**s menu, select **Geometries > Center Mark**.
2. Select the center of circular edges.
3. To finish, select **Done**.

Intersection mark

Intersection marks are cross-shaped annotations that indicate where two non-parallel lines intersect. You can also mark these points as references for dimensioning.

To add an intersection mark:

1. From the **2D Drawings** menu, select **Geometries > Intersection Mark**.
2. Select two non-parallel lines.
3. To finish, select **Done**.

Video walkthrough

Shapr3D Manual - Add geometries to drawings | 2D Drawings

Note

You can annotate parts of your 2D Drawing with the **Note** option.

To add an annotation:

1. From the **2D Drawings** menu, select **Note**.
2. Select an item to add an annotation, or select an empty space to add a note.
3. Enter your annotation in the textbox that appears.
   Note: The character limit is 1200.
4. Drag and drop the notes anywhere you want.
5. To finish, select **Done**.

To edit an annotation:

1. Double-tap or double-click on the text.
2. Enter your new annotation.
3. To finish, select an empty area.

To delete an annotation, select it and press the **Backspace** or **Delete** key, or select **Delete Selection.**

Video walkthrough

Shapr3D Manual - Add notes to drawings | 2D Drawings

Image

You can add images to your 2D Drawings such as screenshots of your visualized model, logos, or any kind of image that helps communicate your project.

To add an image, follow these steps:

1. From the **2D Drawings** menu, select **Add Image**.
2. Select the image you want to import > **Open**.
3. Use the guide points to adjust the size of your image.
4. Drag and drop your image anywhere you want.
5. To finish, select an empty area in the sheet.

To delete an image, select it and press the **Backspace** or **Delete** key, or select **Delete Selection.**

Video walkthrough

Shapr3D Manual - Add images to drawings | 2D Drawings

Sketch menu

Sketching tools
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Automatic line/arc

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The automatic **Line/Arc** tool is a feature specific to using a pen. In **Automatic** mode, Shapr3D automatically switches your sketch to a line or arc depending on your pen gesture.

To start sketching using automatic **Line/Arc**:

1. Start drawing a line or arc. Notice that Shapr3D automatically enters sketch mode and detects what you're drawing.
2. To switch from arc to line and vice versa while sketching, wiggle your pen.

If you want to override the automatic behavior, follow these steps:

1. Go to **Sketch** > **Line/Arc**.
   For other methods of accessing this tool, visit Accessing tools.
2. Tap the **Line Type** menu below **Line/Arc**.
3. Change **Automatic** to **Line** or **Arc.**
4. Use your pen to draw a line or arc.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

Video walkthrough

Shapr3D Manual - Drawing lines and arcs with a pen | Sketching

Read more

- Accessing tools – Understand how tools appear based on what you select in the modeling space.
- Selecting geometry – Master selection techniques for edges, faces, sketches, and bodies.
- Defining and calculating numerical values – Learn how to apply dimensions, enter precise values, and use formulas.
- Keyboard shortcuts, gestures, and hotkeys – View keyboard shortcuts you can use to access tools and speed up actions.
- Variables and expressions – Set up reusable values and expressions to control your design parametrically.

Line

Use the **Line** tool under the **Sketch** menu to create lines.

To create lines using specific input devices, expand the sections below.

Keyboard and mouse/trackpad

By default, you can create connecting lines through a series of endpoints. Follow these steps to learn how:

1. Define your sketch plane.
2. Select **Sketch** > **Line**.
   For other methods of accessing this tool, visit Accessing tools.
3. To draw a line, click to add an endpoint.
4. Drag your pointer to construct your line.
5. Click to add another endpoint.
6. To draw connecting lines, continue to click and drag your pointer to add more endpoints.
7. To finish drawing lines you can do either of the following:
   - Press **Escape** to cancel or exit an active endpoint.
   - Press **Enter** to set your final endpoint.
     Note: Your drawing will automatically finish if you draw a closed shape by connecting the last endpoint to your initial endpoint.

Touch and pen

1. Define your sketch plane.
2. Go to **Sketch** > **Line/Arc**.
   For other methods of accessing this tool, visit Accessing tools.
3. Tap the **Line Type** menu below **Line/Arc**.
4. Select **Line**.
5. Use your pencil to start drawing a line, just like you would on paper.
6. To finish the line, lift your pencil.
7. To draw connecting lines, start drawing from the endpoint of the previous line.
8. endpoint to your initial endpoint.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

Video walkthrough

Shapr3D Manual - Drawing lines | Sketching

Read more

- Accessing tools – Understand how tools appear based on what you select in the modeling space.
- Selecting geometry – Master selection techniques for edges, faces, sketches, and bodies.
- Defining and calculating numerical values – Learn how to apply dimensions, enter precise values, and use formulas.
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Arc

Use the **Arc** tool under the **Sketch** menu to create arcs.

Keyboard and mouse/trackpad

1. Select **Sketch** > **Arc**.
2. Click and drag your pointer to add two endpoints.
3. Drag your pointer to adjust the curve of the arc.
4. Click to finalize the curve.
5. To finish, press **Escape**.
6. To exit sketching, select **Exit Sketching**.

Touch and pen

1. Define your sketch plane, go to **Sketch** > **Line/Arc**.
   For other methods of accessing this tool, visit Accessing tools.
2. Tap the **Line Type** menu below **Line/Arc**.
3. Select **Arc**.
4. Use your pen to draw an arc.
5. To complete the **Arc** tool, select outside of the sketch.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

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Spline

The **Spline** tool creates a curve by marking specific points of curvature. This creates a polynomial curve, or curve with multiple points. With splines, you can construct curves and create organic shapes. Splines have two different types:

- **Fit Point Splines**
- **Control Point Splines**

Fit Point Splines

Use **Fit Point Splines** to create a curve by marking the specific location of each point through which the curve should pass. Fit Point Splines give you more flexibility to set the spline path, but controlling the whole curve can be challenging because any change to a point impacts other parts of the spline.

Keyboard and mouse/trackpad

To create Fit Point Splines, follow these steps:

1. Define your sketch plane, go to **Sketch** > **Spline**, then choose **Fit**.
   For other methods of accessing this tool, visit Accessing tools.
2. Draw a line segment, then click to add points through which the curve should pass.
3. Continue to add points to define the shape of the curve segments.
4. To finish, press **Escape** or **Backspace**.
5. To move the curve points, exit **Spline** by pressing **Escape**, then click and drag the points.
6. To adjust the curvature at a point, select it and drag the handles.
7. To add points to the spline, secondary click (right-click) on a curve to open the context menu, and select **New Spline Point**.
8. To remove a point, select it and then click **Delete**.

Touch and pen

To create Fit Point Splines, follow these steps:

1. Define your sketch plane, go to **Sketch** > **Spline**, then choose **Fit**.
   For other methods of accessing this tool, visit Accessing tools.
2. Draw a curve segment, then lift the pen.
3. Draw more connecting curve segments.
4. To move the curve points, select and drag them directly.
5. To adjust the curvature at a point, drag the handles.
6. To add points to the spline, select the curve and long-tap it.
7. To remove a point, select it and then tap **Delete**.

Control Point Splines

Use **Control Point Splines** to draw and shape a curve indirectly. As you draw each section, a polyline appears to guide the curve. Each point that you mark is a vertex of an imaginary polyline that you can click/tap and move to adjust the shape of the curve. This approach gives you less direct manipulation, but more control over the full curve's smoothness.

Keyboard and trackpad

To create Control Point Splines, follow these steps:

1. Define your sketch plane, go to **Sketch** > **Spline**, then choose **Control**.
   For other methods of accessing this tool, visit Accessing tools.
2. Draw connecting lines. This creates a polyline that guides the shape of the curve.
3. To finish, press **Escape** or **Backspace**.
4. To move the curve points, exit **Spline** by pressing **Escape**, then click and drag the control points, not the curve itself.
5. To add a control point, secondary click (right-click) on a curve to open the context menu, and select **New Spline Point**.
6. To remove a point, select it and click **Delete**.

Touch and pen

To create Control Point Splines, follow these steps:

1. Define your sketch plane, go to **Sketch** > **Spline**, then choose **Control**.
   For other methods of accessing this tool, visit Accessing tools.
2. Draw connecting lines. This creates a polyline that guides the shape of the curve.
3. To adjust the curve, select and drag the control points, not the curve itself.
4. To add a control point, select the curve and long-tap it.
5. To remove a point, select it and tap **Delete**.

> ### Tip
>
> Constraints allow you to break or join splines at any point. To break or join splines at any point, select it and then click the **Break/Join** badge. To connect straight lines with a tangent curve, make sure the spline's control polyline is in line with the neighboring lines. This will create the curve tangent at these points.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

Video walkthrough

Shapr3D Manual - Drawing splines | Sketching

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Rectangle

Use the **Rectangle** tool under the **Sketch** menu to create rectangles.

Rectangle types can either be: **Center**, **Diagonal**, or **Three-point**.

To select the type of rectangle you want to create, follow these steps:

1. Define your sketch plane, go to **Sketch** > **Rectangle**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the menu below **Rectangle** to access the **Rectangle type** options.
   - **Center rectangle** – A center rectangle is defined by its center point. When you select somewhere on the grid, that point becomes the center of the rectangle (and an anchor point, which is indicated by the X-mark) and you can drag the shape to create the sides of the rectangle.
   - **Diagonal rectangle** – A diagonal rectangle is defined by its two opposite corners. When you select somewhere on the grid, you need to drag an imaginary diagonal line. This line becomes the diagonal length of the rectangle, and the initial point becomes the anchor point.
   - **Three-point rectangle** – A three-point rectangle is defined by three points by first drawing a baseline, and then drawing the height. The remaining sides of the rectangle are automatically created based on the two sides, or three points you specified. The first point you create becomes the anchor point.
3. Select the type of rectangle you want to create.

Keyboard and mouse/trackpad

To create a **Center Rectangle**:

1. Click to place the center of the rectangle
2. Drag your cursor and click to place an outer vertex for the rectangle.

To create a **Diagonal Rectangle** :

1. Click to place the starting diagonal point of the rectangle.
2. Drag your cursor diagonally and click to place an outer vertex for the rectangle.

To create a **Three-Point Rectangle**:

1. Click to place the starting point of the rectangle, then move your pointer to draw a line. This will be the length of your baseline. Click to finalize the length of the baseline.
2. Move your pointer perpendicular to the baseline to define the shape of the rectangle. Click to finalize the shape of the rectangle

Touch and pen

To create a **Center Rectangle**:

Select a point on the grid for the center of the rectangle and drag to position an outer vertex

To create a **Diagonal Rectangle** :

Select a point on the grid for the starting diagonal point of the rectangle and drag diagonally to position an outer vertex.

To create a **Three-Point Rectangle** :

1. Select a point on the grid for the starting point of the rectangle, then drag your pen to create a line. This will be the length of your baseline.
2. Drag your pen perpendicular to your baseline to define the shape of the rectangle.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

Video walkthrough

Shapr3D Manual - Drawing rectangles | Sketching

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Circle

The **Circle** tool creates a closed sketch profile with a constant radius.

To create a circle:

1. Define your sketch plane, go to **Sketch** > **Circle.**
   For other methods of accessing this tool, visit Accessing tools.
2. Place the center point and diameter of the circle in the sketch.
   - **Touch and pen:** Select a point on the grid for the center of the circle and drag to position the circumference.
   - **Keyboard and mouse/trackpad:** Click to place the center of the circle, then move your pointer to define the diameter. Click to finalize the diameter.
     Note: You can specify the value for the circle annotation by entering it on the keyboard before placing the diameter.

When drawing the circle or selecting an existing circle, you will see either a radius or diameter dimension label, depending on your **Circular Annotations** setting.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

Video walkthrough

Shapr3D Manual - Drawing circles

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Ellipse

The **Ellipse** tool creates a closed sketch profile that is defined by a major and minor axis.

To create an ellipse:

1. Define your sketch plane, go to **Sketch** > **Ellipse.**
   For other methods of accessing this tool, visit Accessing tools.
2. Place the center point and circumference of the ellipse in the sketch.
   - **Touch and pen:** Select a point on the grid for the center of the ellipse and drag to position the circumference.
   - **Keyboard and mouse/trackpad:**
     1. Click to place a point, then move your pointer to draw a line. This will be the length of your first axis. Click to finalize the diameter of the first axis.
     2. Move your pointer perpendicular to your first axis to define the second axis. Click to finalize the diameter of the second axis.

     Note: You can specify the values for the major and minor axes before placing the endpoints by typing the values using a keyboard.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
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Video walkthrough

Shapr3D Manual - Drawing ellipses | Sketching

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Polygon

The **Polygon** tool creates a closed sketch profile with a specified number of equal sides. There are several pre-defined polygon types available. Once a polygon is created, you can customize the number of sides.

To create a **Polygon**:

1. Define your sketch plane, go to **Sketch** > **Polygon.**
   For other methods of accessing this tool, visit Accessing tools.
2. Select the menu below **Polygon** to access the **Polygon type** options.
3. Select the type of polygon you want to create. The types of pre-defined polygons include:
   - **Triangle**
   - **Pentagon**
   - **Hexagon**
   - **Octagon**

     Note: You can adjust the number of sides for the polygon after you place it in the sketch.
4. Place the center point and a vertex of the polygon in the sketch. For using specific input devices, click the sections below.
   - **Touch and pen**: Select a point on the grid for the center of the polygon and drag to position an outer vertex.
   - **Keyboard and mouse/trackpad**: Click to place the center of the polygon, then move your pointer to define the radius. Click to place the outer vertex.

     Note: You can specify the radius for the polygon before placing the vertex by typing the value using a keyboard.
5. Use the dimension labels to adjust the number of polygon sides and define the radius.

   Note: The label for adjusting the number of sides will only be available immediately after creating the polygon. If you need to adjust this parameter, select the sides label first. Once another selection is made, the sides label will disappear.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
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Video walkthrough

Shapr3D Manual - Drawing polygons | Sketching

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Offset Edge (Sketch)

The **Offset Edge** tool creates new sketch elements that are offset from existing elements by a specified distance. You can use the **Offset Edge** tool within a sketch, with sketch profiles, or with the edges and faces of a 3D body.

Note: For information on using Offset Edge with sketch profiles and edges of 3D bodies, visit Offset Edge (3D).

To offset from a sketch element:

1. Within an existing sketch, go to **Sketch** > **Offset Edge.**
   For other methods of accessing this tool, visit Accessing tools.
2. Select **Type** to access the **Offset Edge Type** menu.
3. Choose an offset type:
   - **Single** – Offsets a single edge from a sketch element.
   - **Chain** – Offsets a chain of edges from a sketch element.
4. Select the sketch element you want to offset.
5. Use the gizmo to specify the offset direction and distance.
   Note: If you select a sketch item that is shared by multiple loops, arrows will be displayed for each loop. Select the appropriate arrow for the loop you want to offset.
6. To finish, select **Exit Sketching**.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

Video walkthrough

Shapr3D Manual - Offsetting sketches | Sketching

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Move/Rotate (Sketch)

Use the **Move/Rotate** tool in sketch mode to move or rotate sketch elements. This tool also includes a **Copy** function that allows for selected elements to be duplicated. The **Move/Rotate** tool uses the gizmo, which provides arrows and tiles for directional control, can be aligned with other geometry, and includes dimension labels for defining precise movements.

Note: To move, rotate, and copy bodies and sketch profiles in the 3D environment, see **Move/Rotate (3D)**.

Using the Move/Rotate tool

To use the **Move/Rotate** tool to move or rotate sketch elements:

1. Within an existing sketch, go to **Sketch** > **Move/Rotate**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the sketch element(s) you want to move.
   Note: To select a group of connected sketch elements, double-click or double-tap one of the sketch elements in the group.
3. If you want to move or rotate a copy of your selection rather than the original, select the **Copy** badge.
   Note: The **Copy** badge appears as blue when selected .
4. Use the gizmo to specify the new location.
5. Complete the **Move/Rotate** tool by selecting an empty area of the grid, or pressing **Escape**.

> ### Tip
>
> For simple movements within a sketch, you can select sketch elements and drag them to a new position. For more complex movements, the **Move/Rotate** tool includes the additional functionality provided by the gizmo.

Moving planes

While in sketching mode, you can move your sketch along the planes using the **Move/Rotate** tool.

To move your sketch along the planes:

1. Set the view to default.
   To learn about the different ways to change the view, visit Orientation Cube or Views and Appearance.
2. Select **Move/Rotate**.
3. Drag the tiles in the gizmo center to move your sketch along the planes.

Moving and adjusting selections

While in sketch mode, you can directly move and adjust selected sketch elements without using the **Move/Rotate** tool. This specific functionality is useful for adjusting certain sketch elements and constraints.

To move and adjust your selections, simply select sketch profiles, edges, or points, then drag or adjust them accordingly.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

Video walkthrough

Shapr3D Manual - Moving and rotating sketches | Sketching

> **Tip**
>
> To find more video tutorials, visit our Shapr3D Manual YouTube Playlist.

> **Read more**
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Pattern (Sketch)

The **Pattern** tool within the **Sketch** menu creates patterns of sketch elements such as lines, arcs, and curves. Patterns are multiple copies of selected elements, evenly spaced in the modeling space.

The **Pattern** tool within in sketch mode can generate two types of patterns:

- Linear pattern
- Circular pattern

Note: For information on patterning 3D bodies or sketch profiles in 3D, visit Pattern (3D).

> **Tip**
>
> Sketch patterns include a default pattern constraint feature that maintains the relationship between the pattern instances. For more information on this constraint, visit Sketch pattern constraint.

Linear pattern

A linear sketch pattern can be distributed along one or two straight axes. Follow these steps to create a linear pattern within a sketch:

1. Within an existing sketch, go to **Sketch** > **Pattern.**
   For other methods of accessing this tool, visit Accessing tools.
2. Select the menu below **Pattern** to access the **Pattern type** menu.
3. Set the pattern type to **Linear**.
4. Select the sketch elements that you want to pattern.
5. Customize the pattern using the pattern control badges:
   - **Pattern Definition**: Use the first pattern control badge to select how you want to define the pattern. The options include:
     - **Total Distance [Total]**: Select this option to define the pattern using an overall dimension. Copies of the selected sketch elements will be equally spaced across the dimension specified.
     - **Spacing Distance [Spacing]**: Select this option to define the pattern by specifying the distance between each copied instance.
   - **Quantity**: Use this pattern control badge to enter the number of copies required for the pattern.
6. Drag a gizmo arrow in the direction you wish to create the pattern.
7. Select the dimension label beside **Total** or **Spacing** to enter a value.
8. To define a second direction for the pattern, repeat steps 5 & 6 with the second gizmo arrow.
9. Complete the **Pattern** tool by selecting an empty area of the grid, or pressing **Escape**.

Circular pattern

A circular sketch pattern distributes copies of selected elements around a specified center point. Follow these steps to create a circular pattern within a sketch:

1. Within an existing sketch, go to **Sketch** > **Pattern.**
   For other methods of accessing this tool, visit Accessing tools.
2. Select the menu below **Pattern** to access the **Pattern type** menu.
3. Set the pattern type to **Circular**.
4. Select the sketch elements that you want to pattern.
5. Move the gizmo center to where you want the pattern center to be.
6. Customize the pattern using the pattern control badges:
   - **Pattern Definition**: Use the first pattern control badge to select how you want to define the pattern. The options include:
     - **Total Angle [Total]**: Select this option to define a pattern using an overall angle. Copies of the selected sketch elements will be equally spaced across the angle specified.
     - **Spacing Angle [Spacing]**: Select this option to define the pattern by specifying the angle between each copied instance.
   - **Quantity**: Use this pattern control badge required for the pattern.
   - **Circular Orientation**: Use this pattern control badge to define how the pattern instances will be rotated. The options include:
     - **Uniform**: Select this option to maintain the original item's orientation for each copy.
     - **Rotated**: Select this option to rotate each copy's orientation relative to the gizmo center.
7. Drag a gizmo arrow in a circular motion to create the pattern.
8. Select the dimension label beside **Total** or **Spacing** to enter a value.
9. Complete the **Pattern** tool by selecting an empty area of the grid, or pressing **Escape**.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

Video walkthrough

Shapr3D Manual - Creating sketch patterns | Sketching

Read more

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Text

The **Text** tool lets you add text to your projects. You can add text to a default plane, a face, or a construction plane. Text is created as sketch profiles and can be used for solid features in the same way as any other sketch.

To add text a sketch or model:

1. Determine how you want the text to be added to your model.
    - To add text to a default plane (Front, Top, Right):
        1. Adjust your view to look normal to the desired plane.
        2. Go to **Sketch** > **Text**.
    - To add text to an existing face or construction plane:
        1. Pre-select an existing planar face or construction plane.
        2. In the **Adaptive User Interface** menu at the left, select **Add Text**.
           Note: You may need to select **More** to access the **Add Text** tool.
2. Use the **Add Text** dialog to define the text. You can also check your model to see how the text looks in proportion to your intended destination.
    - Enter the text into the **Content** field.
    - Adjust the **Font**, **Height**, and **Alignment** settings.
      Note: Shapr3D recognizes all fonts installed on your device.
3. Select **Continue**.
4. Position the text using the gizmo.
5. Select **Done**.

Note: Once the **Text** tool is completed, the text can only be edited as sketch elements.

> ### Tip
>
> If you have chosen a font that produces discontinuous sections, you may need to adjust the sketch before the text can be used for a solid feature.

History settings

To manage history settings for the automatic Line/Arc action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – If applicable, click/tap **Edit…** to modify any existing projections to your sketch.
  Note: To learn more about sketch projections, visit Project.

Video walkthrough

Shapr3D Manual - Adding text | Sketching

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Project - Sketches

The Project - Sketches tool from the Sketch menu is another way to access the Sketches projection type in the Project tool. To learn about this tool, go to Project.

Trim

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The **Trim** tool removes sketch segments between two points or from within a boundary.

To use the **Trim** tool:

1. Within an existing sketch, go to **Sketch** > **Trim**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select unwanted sketch segments.
3. Complete the **Trim** tool by selecting an empty area of the grid, or pressing **Escape**.

Video walkthrough

Shapr3D Manual - Trimming sketches | Sketching

Read more

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Delete (Sketch)

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The **Delete** tool lets you delete sketch elements.

To delete a sketch:

1. Within an existing sketch, select the sketch elements you want to delete.
   For other methods of accessing this tool, visit Accessing tools.
2. Select **Delete**.
3. To finish, press **Escape** or select **Exit sketching**.

Video walkthrough

Shapr3D Manual - Deleting sketches | Sketching

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- Defining and calculating numerical values – Learn how to apply dimensions, enter precise values, and use formulas.
- Keyboard shortcuts, gestures, and hotkeys – View keyboard shortcuts you can use to access tools and speed up actions.
- Variables and expressions – Set up reusable values and expressions to control your design parametrically.

Constraints menu

Constraint overview, settings, and types
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Constraints overview

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Constraints are geometric rules that define the relationship between sketch elements on the same plane. They allow you to "lock" your design intent, ensuring that your sketches remain predictable and precise when you make modifications.

Introduction to Constraints

In Shapr3D, Constraints provide control over how sketch elements respond to changes. By defining specific behaviors—such as making two lines parallel or a line tangent to a circle—you dictate the logic of your geometry.

For example, applying a **Horizontal/Vertical** constraint to a line ensures it stays aligned with the sketch axes, even if the connected parts of the model are moved or resized.

When working in a sketch, the constraints menu automatically highlights valid options based on your selected elements. You can apply these manually or enable **Constraint Settings** to have the app automatically create them for you as you draw.

Video walkthrough

Shapr3D Manual - Introduction to constraints | Sketching

Constraints Settings

The **Constraint Settings** option is where you can adjust options related to constraints. You can find this option in the constraints menu opposite the sketch menu whenever you're working in sketch mode.

Auto-constraining

The **Auto-constraining** option can be adjusted to control if constraints are added to your sketch automatically or manually.

When turned off, the only constraints that are automatically created are between connected endpoints or midpoints. To create additional constraints, select sketch elements and choose your preferred constraint from the constraints menu.

When turned on, as you create sketch elements, constraints for connected endpoints and midpoints are automatically created including **Horizontal/Vertical**, **Perpendicular**, **Tangent** (when an arc is created at an endpoint), and **Coincident**.

> **Tip**
>
> Use **Sketch Guide Lines** to align geometry and automate these constraints. For more information on **Sketch Guide Lines**, visit Snapping Options.

Constraint & Locked Dimension Visibility

The **Constraint & Locked Dimension Visibility** setting controls how constraint icons for the active sketch are shown.

- **Always Show Constraints** – When turned on, constraint icons are only shown for selected sketch elements.
- **Always Show Dimensions** – When turned on, all locked dimensions for the active sketch are shown in the modeling space.

Anchored Sketch Entity

The **Anchored Sketch Entity** setting allows you to specify whether the first or last selected sketch entity stays in place when applying constraints, making it easier to control which part of your sketch moves or stays fixed.

- **First Selected** – When you choose this option, all sketch entities will be moved relative to the one you selected first.
- **Last Selected** – When you choose this option, all sketch entities will be moved relative to the one your selected last.

Note: Existing constraints override this setting.

Video walkthrough

Shapr3D Manual - Introduction to constraints | Sketching

Adding and removing constraints

You can add and remove constraints in a sketch by using options in the constraints menu and constraint icons in the modeling space.

Adding constraints

You will add most constraints by using the constraints menu. However, some constraints can be created directly in the modeling space by using drag and drop or the **Lock** icon. To learn more, explore the sections below.

> **Tip**
>
> You can automate some constraint types by turning on the **Auto-constraining** settings. For more information on **Auto-constraining**, visit Constraint Settings.

Using the constraints menu

The constraints menu automatically opens while in sketching mode. This menu adapts to show applicable constraints based on your selections.

To add a constraint from the menu:

1. Choose a plane with the sketch elements you want to constrain.
2. Select the sketch element(s) you want to constrain.
3. Select the constraint type from the menu.

Note: Some constraint types require selecting more than one sketch element. For information on how to select multiple items, see **Selecting geometry**.

Using keyboard shortcuts

Constraints also have keyboard shortcuts, combining the **Shift** key with an additional letter. You can see the shortcuts for each constraint next to its name in the side menu when using Shapr3D with a keyboard. You can use the shortcuts the same way as the menu: select the element(s) you want to constrain and then press the right key combination. For more information about keyboard shortcuts, visit Keyboard shortcuts, gestures, and hotkeys.

Drag and drop

You can create the following constraints using drag and drop within a sketch:

- **Connected points** – Create connected points by dragging one point onto another point.
- **Coincident** – Create a **Coincident** constraint by dragging a point onto a line or curve.
- **Midpoint** – Create a **Midpoint** constraint by dragging a point onto a line's midpoint (indicated in purple)

Note: Points that are connected, coincident, or have a **Midpoint** constraint are shown with a filled center. For more information, see **States of sketch points**.

Lock

The Lock constraint fixes a selected element in its current position. Visit Lock/Unlock to learn more.

Removing constraints

You can remove constraints by using **Disconnect**, **Unlock**, or by selecting and deleting constraint icons in the modeling space.

> **Tip**
>
> You can control how constraint icons are shown in the modeling space by adjusting the Constraint Settings.

Video walkthrough

Shapr3D Manual - Adding and removing constraints | Sketching

Parallel

Parallel lines have the same distance continuously between them. The Parallel constraint creates a parallel relationship between sketch line elements.

To create a **Parallel** constraint,

1. Select two or more sketch lines on the same plane.
2. From the constraints menu, select **Parallel**.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Perpendicular

Perpendicular lines have a right angle (90°) between them. You can use the Perpendicular  constraint between sketch elements.

To create a **Perpendicular** constraint:

1. Select two lines on the same plane.
2. From the constraints menu, select **Perpendicular**.

> **Tip**
>
> Lines do not need to share a common point or intersect with each other to be made perpendicular. Guide curves are shown to indicate a perpendicularity between disconnected lines.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Tangent

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You can use the Tangent constraint to create a tangent relationship between sketch elements. A line is tangent to a curve and perpendicular to the radius when it touches it at exactly one point. A line that is tangent to an arc (or circle) will be perpendicular to the arc's radius. Arcs and circles can also be tangent to each other. When curves are tangent to each other, they touch at exactly one point.

To create a **Tangent** constraint:

1. On the same sketch plane, select two sketch elements, either a line and a curve, or two curves.
2. Select two sketch elements, either a line and a curve, or two curves.
3. From the constraints menu, select **Tangent**.

> **Tip**
>
> Tangent elements do not need to be in contact with each other. When tangent elements are separated, guide curves are shown to indicate where the tangency applies.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Coincident

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When geometric elements are coincident, it means they are overlapping. You can use the **Coincident**  constraint to make sketch endpoints coincident with other elements to make them connected.

To create a **Coincident** constraint:

1. On the same sketch plane, select an endpoint and another element.
2. Select **Coincident** from the constraints menu or drag an endpoint onto the element you want to connect to.
   Note: Endpoints can be made **Coincident** to other endpoints, lines, curves, and edges.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Read more

Endpoints that are connected to elements with a Coincident constraint are shown with a filled center. For more information, visit States of sketch points.

Midpoint

You can use the Midpoint  constraint to connect an endpoint with the center of a line. This constraint ensures that the connected endpoint remains at the center of the line even if the length of the line changes.

To create a **Midpoint** constraint:

1. On the same plane, select an endpoint and a line.
2. Select **Midpoint** from the constraints menu or drag an endpoint onto the center of a line.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Read more

Endpoints that are connected to elements with a Midpoint constraint are shown with a filled center. For more information, visit States of sketch points.

Concentric

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Concentric arcs share the same center point. You can use the Concentric  constraint to create a concentric relationship between sketch arc elements.

To create a **Concentric** constraint:

1. On the same plane, select arc or circle elements.
2. From the constraints menu, select **Concentric**.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Horizontal/Vertical

You can use the Horizontal/Vertical constraint to create a horizontal or vertical relationship between linear sketch elements. Horizontal and vertical directions in a sketch are aligned with the axes of the sketch plane.

To create a **Horizontal/Vertical** constraint:

1. Select a single or multiple lines on the same sketch plane.
2. From the constraints menu, select **Horizontal/Vertical**.
   Note: The line(s) will be aligned with the axis closest to their original direction.

You can automatically create **Horizontal/Vertical** constraints while sketching when the constraint setting for **Auto-constraining** is turned on. For more information, visit **Constraint settings**.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Equal

The Equal = constraint creates an equal length for selected lines and an equal radius for selected arcs and circles.

To create an **Equal** constraint:

1. Select similar elements on the same sketch plane, such as multiple lines or multiple arcs.
2. Select **Equal** from the constraints menu.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Symmetry

Symmetrical elements lie on opposite sides of an axis of symmetry and behave as mirror images of each other. You can use the Symmetry  constraint to create a symmetrical relationship between sketch elements.

To create a **Symmetry** constraint:

1. On the same sketch plane, select two similar elements, such as two arcs or two lines.
2. Select **Symmetry** from the constraints menu.
3. Select a line or an edge for the axis of symmetry.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Disconnect

The **Disconnect** constraint is used to break the connection between connected points.

To use **Disconnect**:

1. Select a connected line, arc, endpoint, or center point on the same sketch plane.
2. From the constraints menu, select **Disconnect** or directly select for sketch points.
   Note: Disconnect will delete any Coincident or Midpoint constraints for a selected point.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Read more

Connected sketch points are shown with a filled center. For more information, visit States of sketch points.

Lock/Unlock

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The Lock/Unlock constraint fixes a selected sketch element in its current position and the Unlock constraint removes any locked constraint.

To use the **Lock** or **Unlock** constraint:

1. Select an endpoint or center point in the sketch.
2. From the constraints menu, select **Lock** or **Unlock** .

   Note: You can also directly select or on sketch points.

Video walkthrough

Shapr3D Manual - Types of constraints | Sketching

Read more

Locked sketch points are shown as a solid blue color. For more information, visit States of sketch points.

Make construction

Use **Make Construction** to convert your sketch elements into construction geometry.

Construction sketch geometry is indicated with a dashed line.

Construction geometry is used for reference in a sketch. Some examples of sketch elements used for reference include:

- A sketch line you create as an axis of symmetry.
- A sketch line you create as an axis for using **Revolve**.
- Sketch geometry that you use to align other sketch elements, such as a horizontal line between arc centers.
- Sketch geometry that you will reference in a future sketch or feature.
- Sketch geometry that you will not use for 3D geometry.

Construction geometry will not create a filled sketch.

There is no requirement for reference sketch elements to be converted to construction geometry, but it can be a good way to keep your sketches clean and organized.

To use the **Make Construction** option:

1. Select one or multiple existing sketch elements.
2. Click **Make Construction** from the constraints menu.

You can convert construction geometry back to regular sketch elements by selecting it and clicking **Make Regular** from the constraints menu.

> **Tip**
>
> When exporting projects, construction geometry is only visible in the Shapr3D native format. For exported sketches, X_T, STEP, or IGES files, all sketch elements, including construction lines, are skipped.

Video walkthrough

Shapr3D Manual - Making construction geometries | Sketching

Insert menu

Add variables, and import images, files, or projects into your workspace
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Variables and Expressions

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You can use variables and expressions in Shapr3D to define and calculate values dynamically in input fields. Expressions let you perform calculations directly within numerical fields, while variables allow you to assign names to values and reuse them across multiple parameters. This makes it easy to adjust dimensions, constraints, and transformations without manually updating each value.

Note: Expressions use decimal format by default. Fractional imperial input can be written using supported foot (') and inch (") symbols or by using addition (e.g., 2 ft + (1 + 1/2) in). See Unit rules for details.

Variables

Variables let you assign names to numerical values, making it easier to reuse them across multiple parameters. When a variable is updated, all linked values automatically adjust, streamlining modifications without manually updating each instance.

Created variables appear in the **History Sidebar**, where you can rename them and update their references.

Variable naming rules

When you create variables, follow these guidelines:

- Variable names can include English alphanumeric characters (A–Z, a–z), underscores (_), and digits (0–9).
- Variable names must not start with a digit.
- Variable names must be no longer than 100 characters.
- Variable names must not start with double underscores (starting with a single underscore is allowed).

Create and define a variable

You can create a variable in two ways: from the Variables panel or directly from an input field. Once created, variables are automatically added to the History sidebar, where you can rename them, update their values, and manage their references.

From the Variables Panel

You can create variables from the Variables panel. The panel stays open, allowing you to define multiple variables without reopening it each time.

To create variables from the Variables panel:

1. From the main menu, select **Add** > **Variables**.
2. Choose the variable type:
    - **Length** – Distance-based values (e.g., mm, cm, in, ft).

    - **Angle** – Angular values (e.g., degrees).
    - **Number** – Numerical values for general calculations.
3. In the **left field**, enter a variable name, and in the **right field**, enter the numerical value to assign, following the format **Variable Name = Numerical Value**.
4. To finish, select **Create Variable**.

From the input field

You can create a variable directly from any input field to store and reuse values across multiple parameters.

To create a variable from the input field:

1. Open the input field where you want to define a value.
2. Enter a numerical value for the variable.
3. Select **Insert Variable** beside the input field.
4. Choose "**Create [variable_default_name] = [value]**" to confirm the variable creation.
    - **Example**: Entering 4 for a length will display "Create length1 = 4".
5. Check the **History Sidebar** to see the created variable.

History updates

Created variables appear in the History Sidebar with default variable names. From there, you can rename variables and update their references as needed.

To rename and update references for a variable:

1. Open the History Sidebar.
2. Locate the variable you want to update.
3. Right-click or long-tap on the variable name.
4. Select **Rename and Update References**.
5. Enter a new variable name and confirm.

To update only the value:

1. Tap or hover over the value in the History Sidebar.
2. Enter a new value in the input field.

Using variables in expressions

Variables you create and define can be used in expressions to replace fixed values, making it easier to adjust dimensions and constraints dynamically. When updated, all linked values adjust automatically, eliminating manual recalculations. Variables also work with arithmetic operations and unit conversions like regular numerical values, making them useful for parametric design.

Add a variable to an expression

To add variables to expressions:

1. Select **Insert Variable**  beside the input field.
2. Choose the variable from the list to insert it into the expression.

Examples of variables in expressions

Scenario	Expression	Defined Variable	Result
Adjusting box height based on width	box_height = box_width * 2	box_width = 50 mm	box_height = 100 mm
Centering a hole inside a circular part	hole_offset = outer_diameter / 2	outer_diameter = 40 mm	hole_offset = 20 mm
Scaling a feature dynamically	scaled_length = base_length * 1.5	base_length = 80 mm	scaled_length = 120 mm

Expressions

You can use expressions to perform calculations directly within input fields to define numerical values dynamically. Expressions support basic arithmetic operations and can be combined with variables to automate design modifications.

Entering expressions

Expressions can be used in any numerical input field that appears when defining values, including distances, angles, quantities, sketch dimension constraints, and transformations.

To enter an expression:

1. Open the input field.
2. Enter the expression directly in the input field.
3. Press **Enter** to apply the calculated value.
4. To use a variable, select the variable icon to insert it into the expression.

If the expression is valid, the result will be applied. If invalid, an error message will be displayed.

Supported Operations

Expressions support basic arithmetic operations and variables can also be used within expressions for dynamic calculations:

- Addition (+): 10 + 5 → 15
- Subtraction (-): 20 - 4 → 16
- Multiplication (*): 8 * 3 → 24
- Division (/): 12 / 3 → 4
- Parentheses (): (5 + 3) * 2 → 16 (used to control order of operations)

Supported functions

Shapr3D supports a wide range of built-in functions that can be used within expressions to perform mathematical and trigonometric operations as shown in the table below. These functions make it easier to define dynamic relationships and automate calculations across your design.

Before using functions, review these rules:

- Function names are case-sensitive.
- Functions are evaluated using Parasolid's precision tolerance of **1e-8**. For example, ceil(1e-10) will return 0, since the value is interpreted as zero within that range. Similarly, functions like floor() and ceil() may round differently depending on how close values are to whole numbers under this precision.

To learn how values are calculated and prioritized, check the Order of operations and Handling errors sections.

Function	Description	Aliases
sqrt(n)	Returns the square root of n	
sign(n)	Returns 1 if positive, 0 if zero, -1 if negative	sgn
floor(n)	Largest integer not greater than n	
ceil(n)	Smallest integer not less than n	ceiling
round(n)	Nearest integer to n	
abs(n)	Returns the absolute value of n	absolute
mod(a, b)	Remainder of a divided by b	remainder
min(a, b)	Returns the lesser of a or b	minimum
max(a, b)	Returns the greater of a or b	maximum
avg(a, b)	Returns the average of a and b	average, mean
sin(x)	Sine of angle x	
cos(x)	Cosine of angle x	
tan(x)	Tangent of angle x	tg
cotan(x)	Cotangent of angle x	cot, ctg
sec(x)	Secant of angle x	
cosec(x)	Cosecant of angle x	csc
arcsin(x)	Arcsine of x	asin

arccos(x)	Arccosine of x	acos
arctan(x)	Arctangent of x	atan
arctan2(x, y)	2-argument arctangent of y and x	atan2
arccotan(x)	Arccotangent of x	acot, actg
arcsec(x)	Arcsecant of x	asec
arccosec(x)	Arccosecant of x	acsc
pi()	Returns the value of π as a scalar number	
radians(x)	Returns the radians in angle x as a scalar number	

> **Important**
>
> For complex expressions—such as those using operators, variables, functions, or units—use decimal points (.) for numbers. Commas aren't supported in function calls or unit definitions.

Supported Units

Expressions and variables allow unit conversions by specifying different units within calculations.

Examples:

- 5 mm + 2 cm → 25 mm
- 3 in * 2 → 6 in
- 10 mm / 2 → 5 mm

If no unit is specified, the workspace default unit is applied.

Length units

- **Millimeter**: mm, millimeter(s)
- **Centimeter**: cm, centimeter(s)
- **Meter**: m, meter(s)
- **Inch**: in, inch(es), "
- **Foot**: ft, foot, feet, '

Angle units

- **Degrees**: deg, degree(s)

Unit rules

- If you mix different length units (e.g., 10 mm + 1 in), the result is automatically converted based on the active workspace unit.
- Angle units must be explicitly defined (e.g., 30 deg instead of 30°).
- You can use expressions to write imperial fractional values (e.g., 2 ft + (1 + 1/2) in).
  Note: Expressions do not support fractional input (1 1/2 in is invalid; instead, use 1.5 in).
- Fractional foot/inch input is supported when using the ' (feet) and " (inches) symbols, but must follow these rules:
  - Integer inches must always be present if inches are used.
  - Valid forms include:
    - 1' → 1 ft
    - 1" → 1 in
    - 1 1/2" → (1 + 1 / 2) in
    - 1' 1" → 1 ft + 1 in
    - 1' 1 1/2" → 1 ft + (1 + 1 / 2) in
  - Some forms are unsupported or ambiguous:
    - 1/2" translates incorrectly as 1 / (2 in). Use 0 1/2" instead.
    - 1' 1/2" causes a parser error. Use 1' 0 1/2" instead.

Order of Operations

Expressions follow the standard mathematical order:

1. Parentheses ()
2. Multiplication * and Division /
3. Addition + and Subtraction -

Example: 10 + 2 * 3 results in 16, while (10 + 2) * 3 results in 36.

Handling Errors

- If an expression contains invalid characters or incorrect syntax, an error message appears.
- Ensure all expressions use supported operators and follow the correct format.
- Mixed unit types (e.g., 5 mm + 3 deg) are not allowed.

Read more

For basic numerical input and calculations,, visit Defining and calculating numerical values.

Insert Image

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The **Image** option under the **Insert** menu is another way to access **Import**. To learn more about importing images, go to **Import**.

Insert File

[Article link | Table of contents | Back | Next]

The **File** option under the **Insert** menu is another way to access **Import**. To learn more about importing files, go to **Import**.

Insert Project

[Article link | Table of contents | Back | Next]

The **Project** option under the **Insert** menu is another way to access **Import**. To learn more about importing files, go to **Import**.

Construct menu

Custom construction planes and axes for reference geometry
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Understanding construction geometry

[Article link | Table of contents | Back | Next]

Construction geometry refers to geometry that is used for reference in a project. The types of construction geometry are:

- Construction planes
- Construction axes
- Construction sketch elements

Construction geometry is sometimes required to properly position elements of a project. It is often created when there isn't an existing project element to reference at a desired location. Some examples of construction geometry include:

- Creating a Construction Plane at the desired location for a new sketch.
- Creating a Construction Plane to use as a mirroring plane.
- Creating a Construction Axis at the center location for a circular pattern.
- Sketching a horizontal construction line between arc centers to keep them in alignment.

> **Tip**
>
> When exporting projects, construction geometry is only visible in the Shapr3D native format. For exported sketches, X_T, STEP, or IGES files, all sketch elements, including construction lines, are skipped.

Construction Plane

Use the **Construction Plane** tools to create custom reference planes. Some examples of creating construction planes include:

- Creating a plane at the desired location for a new sketch.
- Creating a plane that can be used as a mirroring plane.
- Creating a cutting plane for splitting bodies or making sectioned views.

> **Tip**
>
> Planes are infinite but are represented in the modeling space with a size based on how they are defined. For general information regarding construction geometry, visit Understanding construction geometry.

Creating a Construction Plane

To create a **Construction Plane**:

1. Go to **Construct** > **Plane** and select the specific plane tool you need.
   For other methods of accessing this tool, visit Accessing tools.
2. Select appropriate references for the type of plane you chose (for details, see Construction Plane types).
3. Select **Done**.

> **Tip**
>
> You can use Shapr3D's **adaptive user interface** as a shortcut for creating some construction plane types. To make use of this shortcut, pre-select an element in your project that will be used to define the plane, then select **Add Plane** from the customized menu at the left. After defining the plane, click in an empty area of the modeling space to complete the command.

Construction Plane types

Each construction axis type is available as a separate tool in the Construct menu. When you select a Plane tool, follow the guide at the top of the screen to complete the steps.

Offset Plane

Use the **Offset Plane** construction plane type to create a plane that is offset from an existing planar element. Follow these steps to create this type of plane:

1. Select a face, plane, or sketch profile.
2. Drag the arrows to define the offset distance or use the dimension label to enter a value.
3. To finish, select **Done**.

History settings

To manage history settings for the Construction Plane action, find the step for your specific construction plane offset from face and then view or modify the following:

- **Face** – Click/tap **Edit…** or **Select…** to choose a different face and then select **Done** to finish.
- **Offset** – Enter a new value for your preferred offset distance.
- **Size** – Enter a new value for your preferred construction plane size.

Midplane

Use the **Midplane** tool to create a plane that is centered between two selected planar elements.

To create this type of plane:

1. Select faces, planes, or sketch profiles.
2. To finish, select **Done**.

History settings

To manage history settings for the Construction Plane action, find the step for your specific midplane construction plane and then view or modify the following:

- **Face #1** – Click/tap **Edit…** or **Select…** to choose the first face and then select **Done** to finish.
- **Face #2** – Click/tap **Edit…** or **Select…** to choose the second face and then select **Done** to finish.
- **Size** – Enter a new value for your preferred construction plane size.

Plane Through 3 Points

Use the **Plane Through 3 Points** tool to create a plane that is defined by three selected points.

Follow these steps to create this type of plane:

1. Select existing points or select anywhere in the modeling space to define the three points for the plane.
2. **Optional**: Drag the automatically generated points to reposition the plane. The points will snap to the grid, sketch geometry, or body geometry.
3. To finish, select **Done**.

History settings

To manage history settings for the Construction Plane action, find the step for your specific construction plane through 3 points and then view or modify the following:

- **Point #1** – Click/tap **Edit…** or **Select…** to choose a different first point and then select **Done** to finish.
- **Point #2** – Click/tap **Edit…** or **Select…** to choose a different second point and then select **Done** to finish.
- **Point #3** – Click/tap **Edit…** or **Select…** to choose a different third point and then select **Done** to finish.
- **Size** – Enter a new value for your preferred construction plane size.

Plane on Curve at Point

Use the **Plane on Curve at Point** tool to create a plane that is normal to a curve at a selected point.

Follow these steps to create this type of plane:

1. Select an existing curve, such as a curved edge or sketch curve.
2. Drag the point that is generated to position the plane parameter as desired.
3. To finish, select **Done**.

History settings

To manage history settings for the Construction Plane action, find the step for your specific construction plane perpendicular to curve at point and then view or modify the following:

- **Curve** – Click/tap **Edit…** or **Select…** to choose a different curve and then select **Done** to finish.
- **Point** – Click/tap **Edit…** or **Select…** to choose a different point and then select **Done** to finish.
- **Size** – Enter a new value for your preferred construction plane size.

Plane Perpendicular to Edge at Point

Use the **Plane Perpendicular to Edge at Point** tool to create a plane that is perpendicular to a linear element located at a selected point.

Follow these steps to create this type of plane:

1. Select a linear edge, axis, or sketch line. A reference point is automatically placed at a default position along the edge.
2. **Optional**: To change the location of the point, drag the automatically generated point onto any purple snap-point of your geometry, or select a different snap point.
3. To finish, select **Done**.

History settings

To manage history settings for the Construction Plane action, find the step for your specific construction plane perpendicular to edge at point and then view or modify the following:

- **Edge** – Click/tap **Edit…** or **Select…** to choose a different edge and then select **Done** to finish.
- **Point** – Click/tap **Edit…** or **Select…** to choose a different point and then select **Done** to finish.
- **Size** – Enter a new value for your preferred construction plane size.

Plane Through Edge at Angle

Use the **Plane Through Edge at Angle** tool to create a plane that is rotated about a selected linear element.

Follow these steps to create this type of plane:

1. Select a linear edge, axis, or sketch line. A default reference face and plane preview will automatically display.
2. **Optional**: Select a different face, plane, or sketch profile to change the reference face your angle is measured from.
3. Drag the arrows to define the angle or use the dimension label to enter a value.
4. To finish, select **Done**.

History settings

To manage history settings for the Construction Plane action, find the step for your specific construction plane through edge at angle and then view or modify the following:

- **Edge** – Click/tap **Edit…** or **Select…** to choose a different edge and then select **Done** to finish.
- **Face** – Click/tap **Edit…** or **Select…** to choose a different face and then select **Done** to finish.
- **Angle** – Enter a new value for your preferred angle.
- **Size** – Enter a new value for your preferred construction plane size.

Plane Parallel to Face at Point

Use the **Plane Parallel to Face at Point** tool to create a plane that is offset from an existing planar element and located at a selected point.

Follow these steps to create this type of plane:

1. Select a face, plane, or sketch profile. A reference point is automatically placed at a default position on the surface.
2. **Optional**: To change the point location, drag the automatically generated point onto any purple snap-point of your geometry, or select a different snap point.
3. To finish, select **Done**.

History settings

To manage history settings for the Construction Plane action, find the step for your specific construction plane parallel to face at point and then view or modify the following:

- **Face** – Click/tap **Edit…** or **Select…** to choose a different face and then select **Done** to finish.
- **Point** – Click/tap **Edit…** or **Select…** to choose a different point and then select **Done** to finish.
- **Size** – Enter a new value for your preferred construction plane size.

Video walkthrough

https://www.youtube.com/watch?v=AdkbjwpRQBY

Construction Axis

Use the **Construction Axis** tool to create an axis for reference in your project. Some examples of creating construction axes include:

- Creating an axis for a circular pattern.
- Creating a reference point or snapping point.
- Creating a virtual intersection at a filleted corner.
- Creating an axis for a revolved body.

> **Tip**
>
> Axes are infinite in length but are represented in the design space with a size based on how they are defined. For general information regarding construction geometry, visit Understanding construction geometry.

Creating a Construction Axis

To create a **Construction Axis**:

1. Go to **Construct** > **Axis** and select the specific axis tool you need.
   For other methods of accessing this tool, visit Accessing tools.
2. Select appropriate references for the type of axis you chose (for details, see Construction Axis types).
3. Select **Done**.

> **Tip**
>
> You can use the Adaptive User Interface as a shortcut for creating construction axes. To make use of this shortcut, pre-select an element in your design that will be used to define the axis, then select Add Axis from the customized menu at the left. After defining the axis, click in an empty area of the design space to complete the command.

Construction Axis types

You can define a **Construction Axis** in several different ways using the separate dedicated tools found under the **Construct** > **Axis** menu. The tooltip at the top of the modeling space will provide you with directions on what you need to do for the specific tool you have chosen.

Axis Through 2 Points

Use the **Axis Through 2 Points** tool to create an axis between two selected reference points. You can select sketch points and vertices, or you can select anywhere in the modeling space to define a new point.

Follow these steps to create this type of axis:

1. Select the first point the axis will run through.
2. Select the second point the axis will run through.
3. **Optional**: Drag the automatically generated points to reposition the axis. The points snap to the grid, sketch geometry, or body geometry.
4. To finish, select **Done**.

History settings

To manage history settings for the Construction Axis action, find the step for your specific construction axis through 2 points and then view or modify the following:

- **Point #1** – Click/tap **Edit…** or **Select…** to choose a different first point and then select **Done** to finish.
- **Point #2** – Click/tap **Edit…** or **Select…** to choose a different second point and then select **Done** to finish.

Axis Through 2 Planes

Use the **Axis Through 2 Planes** tool to create an axis at the intersection of two selected planar elements.

To create this type of axis:

1. Select two intersecting planes, faces, or sketch profiles.
2. To finish, select **Done**.

History settings

To manage history settings for the Construction Axis action, find the step for your specific construction axis through 2 planes and then view or modify the following:

- **Plane #1** – Click/tap **Edit…** or **Select…** to choose a different first plane and then select **Done** to finish.
- **Plane #2** – Click/tap **Edit…** or **Select…** to choose a different second plane and then select **Done** to finish.
- **Length** – Enter a new value for your preferred construction axis length.

Axis of Cylinder/Cone

Use the **Axis of Cylinder/Cone** tool to create an axis at the axis of symmetry/rotation of a cylindrical or conical face.

To create this type of axis:

1. Select a cylindrical or conical surface.
2. To finish, select **Done**.

History settings

To manage history settings for the Construction Axis action, find the step for your specific construction axis through cylinder or cone and then view or modify the following:

- **Face** – Click/tap **Edit…** or **Select…** to choose a different face and then select **Done** to finish.
- **Length** – Enter a new value for your preferred construction axis length.

Axis Along Edge

Use the **Axis Along Edge** tool to create an axis at a selected edge or line.

To create this type of axis:

1. Select a linear edge of a body or a sketch line.
2. To finish, select **Done**.

History settings

To manage history settings for the Construction Axis action, find the step for your specific construction axis through edge and then view or modify the following:

- **Edge** – Click/tap **Edit…** or **Select…** to choose a different edge and then select **Done** to finish.

Axis Perpendicular to Face at Point

Use the **Axis Perpendicular to Face at Point** tool to create an axis that is perpendicular to a selected planar element and runs through a reference point. Follow these steps to create this type of axis:

1. Select a face, plane, or sketch profile. A reference point is automatically generated at a default position.
2. **Optional**: To change the reference point location, drag it directly onto any purple snap-point of the geometry, or select a different snap point.
3. To finish, select **Done**.

History settings

To manage history settings for the Construction Axis action, find the step for your specific construction axis perpendicular to face at point and then view or modify the following:

- **Face** – Click/tap **Edit…** or **Select…** to choose a different face and then select **Done** to finish.
- **Point** – Click/tap **Edit…** or **Select…** to choose a different point and then select **Done** to finish.
- **Length** – Enter a new value for your preferred construction axis length.

Video walkthrough

https://www.youtube.com/watch?v=AdkbjwpRQBY

Transform menu

Transformation tools
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Copying items

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Depending on the tool you use, you will see a **Copy** badge or a checkbox option to make a copy of your sketch or body. Choose the appropriate tool based on the summary below:

- **Move/Rotate** – Use this tool to create a copy of a body using the gizmo.
  Note: For creating copies of sketch elements, use **Move/Rotate (Sketch)**.
- **Translate** – Use this tool to create a copy at a selected point.
- **Scale** – Use this tool to create a copy at a different scale.
- **Rotate Around Axis** – Use this tool to create a copy that is rotated about an axis.

For tools that use the **Copy** badge, the color of the badge indicates when it is selected:

- Unselected **Copy** badge: 

- Selected **Copy** badge: 

Video walkthrough

Shapr3D Manual - Copying items | Transform

Move/Rotate (3D)

Use the **Move/Rotate** tool in the 3D environment to move or rotate sketches, edges, faces, and 3D bodies. This tool also includes a **Copy** function that allows for selected elements to be duplicated. The **Move/Rotate** tool uses the gizmo, which provides arrows and tiles for directional control, can be aligned with other geometry, and includes dimension labels for defining precise movements.

Note: To move, rotate and copy sketch elements, visit **Move/Rotate (Sketch).** To move objects by a vector defined by two points, rather than using the gizmo, visit **Translate.**

To use the **Move/Rotate** tool in the 3D environment:

1. Go to **Transform > Move/Rotate.**
   Note: The **Move/Rotate** tool automatically becomes active when a *body* is selected. You can select an entire body by double-clicking any face of the body.
2. Select the elements you want to reposition.
3. If you want to move or rotate a copy of your selection rather than the original, select Copy .
   Note: The **Copy** badge appears as blue when selected Copy .
4. Use the gizmo to specify the new location.
   Note: Use the **Auto-orientation** option at the top left to control whether or not the gizmo aligns with other geometry when moving the gizmo center. This option is only available when accessing the **Move/Rotate** tool from the **Transform** menu.
5. Complete the **Move/Rotate** tool by selecting **Done**.

> **Tip**
>
> You can create new edges to use with the **Move/Rotate** tool by projecting sketch elements onto faces. For more information, visit Project.

History settings

To manage history settings for the Move/Rotate action, find the step for your specific movement/rotation and then view or modify the following:

- **Start Center** – Click/tap **Edit…** or **Select…** to choose a different starting center and then select **Done** to finish.
- **Target Bodies** – Click/tap **Edit…** or **Select…** to choose a different target body and then select **Done** to finish.
- **Target Faces** – Click/tap **Edit…** or **Select…** to choose a different target face and then select **Done** to finish.
- **Target Edges** – Click/tap **Edit…** or **Select…** to choose a different target edge and then select **Done** to finish.
- **Copy** – Toggle the slider to turn on or off the original copy.

Video walkthrough

Shapr3D Manual - Move/Rotate | Transform

Scale

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The **Scale** tool can scale or adjust the size of items. This tool includes a **Copy** option which allows you to create a copy of the item to be scaled while keeping the original element unchanged.

To scale items:

1. Go To **Transform** > **Scale**.
2. Select **Type** to access the **Scale Mode** menu.
3. Choose a uniform type:
    - **Uniform** – Scales an object with a center point adjustment handle in all directions proportionally.
    - **Non-uniform** – Scales an object in any direction with different proportions along axes. You can define a center point for the scale by adjusting the gizmo center location.
4. Select the item you want to scale.
5. If applicable, drag the gizmo center to your preferred location.
   Note: The gizmo center is the center point of the scale.
6. Optional**:** To scale a copy of your selection rather than the original, select **Copy** Copy .

   Note: The **Copy** badge Copy appears as blue when selected .
7. Use the gizmo to adjust the scale of the selection.
8. To finish, select an empty area of the grid.

History settings

To manage history settings for the Scale action, find the step for your specific scale and then view or modify the following:

- **Target Bodies** – Click/tap **Edit…** or **Select…** to choose a different target body and then select **Done** to finish.
- **Target Faces** – Click/tap **Edit…** or **Select…** to choose a different target face and then select **Done** to finish.
- **Target Edges** – Click/tap **Edit…** or **Select…** to choose a different target edge and then select **Done** to finish.
- **Uniform** – Turn on for a uniform scale or turn off for a non-uniform scale.
- **Scale X** – Enter a new scale value for the X-axis.
- **Scale Y** – Enter a new scale value for the Y-axis.
- **Scale Z** – Enter a new scale value for the Z-axis.
- **Copy** – Toggle the slider to turn on or off the original copy.

Video walkthrough

Shapr3D Manual - Scale | Transform

Translate

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The **Translate** tool moves an object by a vector defined by two points. This tool can reposition sketches, sketch profiles, or bodies.

To translate geometry:

1. Go to **Transform** > **Translate**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the sketches, sketch profiles, or bodies you want to translate.
3. Select **Next**.
4. If you want to translate a copy of your selection rather than the original, select **Copy**.
5. Select the start and end points of the desired movement. Both points snap to the grid, edges, or vertices.
6. Select **Done**.

History settings

To manage history settings for the Translate action, find the step for your specific translation and then view or modify the following:

- **Target Bodies** – Click/tap **Edit…** or **Select…** to choose a different target body and then select **Done** to finish.
- **Target Faces** – Click/tap **Edit…** or **Select…** to choose a different target face and then select **Done** to finish.
- **Target Edges** – Click/tap **Edit…** or **Select…** to choose a different target edge and then select **Done** to finish.
- **Start Point** – Click/tap **Edit…** or **Select…** to choose a different start point and then select **Done** to finish.
- **End Point** – Click/tap **Edit…** or **Select…** to choose a different end point and then select **Done** to finish.
- **Copy** – Toggle the slider to turn on or off the original copy.

Video walkthrough

Shapr3D Manual - Translate | Transform

Pattern (3D)

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The **Pattern** tool within the **Transform** menu creates linear or circular patterns of 3D bodies, sketch profiles, or entire sketches. In 3D modeling, a pattern is the arrangement of elements derived from the same source object.

Note: Unlike sketch patterns, patterning bodies will have no connection between them, but patterned bodies are automatically placed in a folder in the items manager to make selections easier. For information on patterning within a sketch, visit Pattern (Sketch).

Linear pattern

A linear pattern can be distributed along one, two, or three straight axes. Follow these steps to create a linear pattern within the 3D environment:

1. Go to **Transform** > **Pattern.**
   For other methods of accessing this tool, visit Accessing tools.
2. Select **Type** to access the **Pattern type** menu.
3. Choose **Linear** as the pattern type.
4. Select the bodies or sketch profiles you want to pattern.
   Note: Double-click/double-tap on a face to select a body.
5. Customize the pattern using the pattern control badges:
   - **Pattern Definition** – Use the first pattern control badge to select how you want to define the pattern. The options include:
     - **Total Distance [Total]** – Select this option to define the pattern using an overall dimension for the pattern. Copies of the selected sketch elements will be equally spaced across the dimension specified.
     - **Spacing Distance [Spacing]** – Select this option to define the pattern by specifying the distance between each copied instance.
   - **Quantity** – Use this pattern control badge to enter the number of copies required for the pattern.
6. Drag a gizmo arrow in the direction you wish to create the pattern.
7. Select the dimension label beside **Total** or **Spacing** to enter a value.
8. To define a second or third direction for the pattern, repeat steps 5 & 6 with the other available gizmo arrows.
9. Complete the **Pattern** tool by selecting **Done**.

Circular pattern

A circular pattern distributes copies of selected elements around a specified center point. Follow these steps to create a circular pattern within the 3D environment:

1. Go to **Transform** > **Pattern**
   For other methods of accessing this tool, visit Accessing tools.
2. Select **Type** to access the **Pattern type** menu.
3. Choose **Circular** as the pattern type.
4. Select the bodies or sketch profiles you want to pattern.
   Note: Double-click/double-tap on a face to select a body.
5. Move the gizmo center to where you want the pattern center to be.
6. Customize the pattern using the pattern control badges:
   - **Pattern Definition** – Use the first pattern control badge to select how you want to define the pattern. The options include:
     - **Total Angle [Total** – Select this option to define a pattern using an overall angle for the pattern. Copies of the selected sketch elements will be equally spaced across the angle specified.
     - **Spacing Angle [Spacing]** – Select this option to define the pattern by specifying the angle between each copied instance.
   - **Quantity** – Use this pattern control badge to enter the number of copies required for the pattern.
   - **Circular Orientation**: Use this pattern control badge to define how the pattern instances will be rotated. The options include:
     - **Uniform** – Select this option to maintain the original item's orientation for each copy.
     - **Rotated** – Select this option to rotate each copy's orientation relative to the gizmo center.
7. Drag a gizmo arrow in a circular motion to create the pattern.
8. Select the dimension label beside **Total** or **Spacing** to enter a value.
9. Complete the **Pattern** tool by selecting **Done.**

History settings

To manage history settings for the Pattern action, find the step for your specific pattern and then view or modify the following:

- **Target Bodies** – Click/tap **Edit…** or **Select…** to choose a different target body and then select **Done** to finish.
- **Target Faces** – Click/tap **Edit…** or **Select…** to choose a different target face and then select **Done** to finish.
- **Definition** – Click/tap the dropdown to choose between **Spacing Distance** and **Total Distance**.
- **Quantity** – Enter your preferred quantity for the pattern.
- **Distance** – Enter your preferred pattern distance.

Video walkthrough

Shapr3D Manual - Pattern | Transform

Rotate Around Axis

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The **Rotate Around Axis** tool can rotate sketches, sketch profiles, edges, faces, or bodies around a selected axis.

How to use the Rotate Around Axis tool

To use the **Rotate Around Axis** tool:

1. Go to **Transform > Rotate Around Axis**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the sketches, sketch profiles, edges, faces, or bodies you want to rotate.
   Select **Next**.
3. Select an axis, line, or edge you want to revolve around.
4. If you want to rotate a copy of your selection rather than the original, select the **Copy** badge.

   Note: The **Copy** badge 

 appears as blue when selected.
5. Use the gizmo to rotate around the axis.
6. Select **Done**.

History settings

To manage history settings for the Rotate Around Axis action, find the step for your specific rotation around an axis and then view or modify the following:

- **Target Bodies** – Click/tap **Edit…** or **Select…** to choose a different target body and then select **Done** to finish.
- **Target Faces** – Click/tap **Edit…** or **Select…** to choose a different target face and then select **Done** to finish.
- **Target Edges** – Click/tap **Edit…** or **Select…** to choose a different target edge and then select **Done** to finish.
- **Axis** – Click/tap **Edit…** or **Select…** to choose a different axis and then select **Done** to finish.
- **Angle** – Enter a new value for your preferred angle.
- **Copy** – Toggle the slider to turn on or off the original copy.

Video walkthrough

Shapr3D Manual - Rotate Around Axis | Transform

Align

You can use the **Align** tool to align 3D bodies.

Making selections

Selections for the **Align** tool include:

- Planar faces
- Spherical faces
- Conical faces
- Sketches
- **Construction planes**
- Construction axes
- Circular edges and sketches
- Linear sketches and edges

Using the Align tool

To use the **Align** tool:

1. Go to **Transform** > **Align**.
   For other methods of accessing this tool, visit Accesing tools.
2. Select the body or bodies that you want to align. This defines the bodies that will be moving in the modeling space.
   Note: If you pre-select a body and select **Align** from the adaptive menu, this step is not necessary.
3. Click **Next**.
4. Select a face or edge from the bodies you want to align. If the body has snap points, they are shown in purple.
5. Select the target that you want to align with. This selection could be any of the valid selections mentioned in the table above. Alternatively, you can select the body's snap point and drag it to a target body's snap point.
6. If necessary, adjust the alignment using the gizmo and the **Flip** badge  . For more information, see Adjusting alignment.
7. Select **Done**.

Adjusting alignment

You can adjust the position of an aligned body using the gizmo and Flip badge. These on-screen tools can help you rotate, offset, and re-orient the body. Here are some examples of how to use these tools to manipulate a bodies alignment: **Note:** For more detailed information, visit **Using the gizmo.**

- Rotating around the gizmo center
- Moving linearly along the gizmo axes
- Moving on a plane
- Flipping 180 degrees

Snapping into alignment

When you adjust alignment using the gizmo, the body will snap into likely positions based on your selections and movements. Examples include:

- Aligned vertices
- Collinear edges
- Coplanar arcs
- Coplanar faces
- Parallel edges

History settings

To manage history settings for the Align action, find the step for your specific alignment and then view or modify the following:

- **Target Body** – Click/tap **Edit…** or **Select…** to choose a different target and then select **Done** to finish.

Video walkthrough

Shapr3D Manual - Align | Transform

Mirror

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You can use the **Mirror** tool to mirror any sketch, face, or body over a selected plane.

To use the **Mirror** tool:

1. Go to **Transform > Mirror**.
   For other methods of accessing this tool, visit Accesing tools.
2. Select the elements that you want to mirror.
   Note: Selected elements to be mirrored will be highlighted in blue and labeled with the Mirror tool badge. You can select the Mirror tool badge to toggle off a selection.
3. Select a base plane, face, sketch profile, axis, sketch line, or construction plane to use as a mirror plane.
   Note: The selected mirror plane will be highlighted in purple and labeled with the target badge. You can select the target badge to toggle off a selection.
4. If you want to remove the original element once the mirror is complete, turn off the **Keep Originals** switch at the left.
5. Select **Done**.

History settings

To manage history settings for the Mirror action, find the step for your specific mirror and then view or modify the following:

- **Target Bodies** – Click/tap **Edit…** or **Select…** to choose a different target body and then select **Done** to finish.
- **Target Faces** – Click/tap **Edit…** or **Select…** to choose a different target face and then select **Done** to finish.
- **Plane** – Click/tap **Edit…** or **Select…** to choose a different plane and then select **Done** to finish.
- **Keep Originals** – Toggle the slider to turn on or off the original body.

Video walkthrough

Shapr3D Manual - Mirror | Transform

Tools menu

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Offset Face

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You can use the **Offset Face** tool to adjust the thickness of faces or bodies by offsetting them. When you offset a face, tangent faces are automatically included in the offset action.

Offsetting faces allows you to:

- Modify thickness by adding or removing material.
- Resize features like holes by expanding or contracting their boundaries.
- Make precise adjustments to isolated parts within patterned designs.

How to use the Offset Face tool

To use the **Offset Face** tool:

1. Access the **Offset Face** tool.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the face or faces you want to offset.
3. Use the gizmo to offset the selected face or enter your preferred offset face distance.
4. **Optional**: If you select only one flat face, you can select the gizmo to choose a Distance Type.
5. Select **Done**.
   Note: When using **Offset Face** from the adaptive menu, select an empty area of the grid to complete the tool.

Offset Face Distance Type

When using the **Offset Face** tool, you can choose a **Distance Type** to determine how the offset is applied. This option is available only when a single face is selected, allowing for precise control over the offset operation.

- **Radius/Diameter**: Adjusts circular faces based on their type. For full circular faces, the diameter is expanded or contracted symmetrically around the center. For partial circular faces, the radius is adjusted.
- **Total**: Sets the overall thickness by offsetting the face and its opposite face (if available) to match the specified total distance.
- **Offset**: Moves the face by a specific distance relative to its original position. When multiple faces are selected, Offset is the only available option.

Comparing tool functions

When familiarizing yourself with Shapr3D tools, **Offset Face** and **Extrude** may appear similar, but each has a unique role:

- **Offset Face**
    - Adjusts the thickness or position of faces, modifying a body by extending or trimming adjacent faces without creating new geometry.
    - Unlike Extrude, Offset Face works on both planar and non-planar faces, making it versatile for refining shapes.
- **Extrude**
    - Creates a 3D body from a 2D sketch or extends extrusions, adding geometry to a feature.
    - Limited to planar faces for creating new bodies.

Practical applications

The **Offset Face** tool offers flexibility across various design scenarios. Here are a few examples of how it can be applied:

- **Adjusting wall thickness**: Offset Face is ideal for modifying wall thickness in a model. For instance, in a hollow box, you can select and offset an inner wall face inward to reduce thickness without affecting the outer dimensions.
- **Customizing patterned features**: When working with patterned bodies or holes, you can use **Offset Face** to modify individual instances within the pattern without altering the others. This is helpful if a specific feature within the pattern needs to differ from the rest.
- **Offsetting adjacent faces**: Offset Face can adjust multiple connected faces at once, applying consistent modifications across adjacent surfaces. This is useful when you need to make uniform adjustments across a feature with several connected faces, such as tapering or expanding a portion of a part.
- **Quickly adjusting prototypes**: In prototyping, Offset Face enables rapid adjustments to dimensions without needing to redraw sketches. This is especially useful for iterating on models when refining designs based on physical tests or feedback.

History settings

To manage history settings for the Offset Face action, find the step for your specific face offset and then view or modify the following:

Multiple face offsets

- **Face** – Click/tap **Edit…** or **Select…** to choose a different face to offset and then select **Done** to finish.
- **Offset Distance** – Enter the offset distance for a single offset or **Offset** distance type.

Single face offset

- **Face** – Click/tap **Edit…** or **Select…** to choose a different face to offset and then select **Done** to finish.
- **Distance Type** – Choose how the offset is applied by selecting one of the following:
    - **Radius/Diameter** – Adjusts the dimensions of circular faces. Selecting this type reveals the following field(s):
        - **Radius/Diameter** – Enter the radius or diameter value for your selection.
          Note: When switching from **Radius/Diameter** to **Offset**, the radius/diameter value is applied as the offset distance, which may lead to unexpected results.
    - **Total** – Sets the overall thickness across the face and its opposite face. Selecting this type reveals the following field(s):
        - **Total Distance** – Enter the total thickness for your selection.
        - **Opposite Face** – Select **Edit** to adjust or change the opposite face.
    - **Offset** – Sets a specific distance from the original face position, adjusting the face independently. Selecting this type reveals the following field(s):
        - **Offset Distance** – Enter the offset distance for a single offset or **Offset** distance type.
          Note: For multiple face selections, **Offset Distance** is the only available option.

Video walkthrough

Shapr3D Manual - Offset Face | Tools

Chamfer/Fillet

You can use the **Chamfer/Fillet** tool to add an angled face or radius at the edges of your model. The type of geometry created from a chamfer or fillet is based on how you manipulate the arrows for the tool and the settings you select. The basic method is to select an edge and use the arrows as described below.

- **Chamfer**: Drag the arrows toward the body.
- **Fillet**: Drag the arrows away from the body.

Chamfer

The Chamfer tool creates beveled edges by cutting off the edge at an angle, rather than rounding it like a fillet. Chamfers are useful for removing sharp edges or for functional and aesthetic purposes in your designs.

How to create a chamfer

To create a chamfer:

1. Go to **Tools** > **Chamfer/Fillet**.
   For other methods of accessing this tool, visit Accessing tools.
2. Click **Chamfer/Fillet** again to choose a chamfer type.
3. Select the edge(s) to chamfer.
4. Drag the arrow(s) to chamfer the selected edges.
5. To finish, select **Done**.
   Note: When using **Chamfer/Fillet** from the adaptive menu, select an empty area of the grid to complete the tool.

Chamfer types

You can create the following chamfer types in Shapr3D:

- **Auto** – Creates an angled face that is setback equal distances from the selected edge. For an edge at the intersection of perpendicular faces (90°), the chamfer produces a 45° bevel. The dimension that you define for a chamfer is the distance the chamfer is set back from the original edge.
- **2-distance** – A chamfer set back at different distances from the selected edge, giving you even more control over how the chamfer is applied. This is useful when the chamfer needs to taper in a specific way.

Fillet

The Fillet tool creates smooth transitions between edges by rounding them off. The default fillet settings result in a face with a circular cross-section that is tangent (**G1** continuity) to adjacent faces. When three or more filleted edges meet at a corner, the default **Corner** setting uses **Rolling Ball**. These settings can be adjusted to fine-tune a fille (see Understanding fillet settings).

How to create a fillet

To create a default circular fillet:

1. Go to **Tools** > **Chamfer/Fillet**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the edg(s) you want to fillet.
3. Use the gizmo to fillet the selected edge(s).
   Note: A radius fillet is created by default. You can change the fillet type in your fillet settings in History.
4. To finish, select **Done**.
   Note: When using **Chamfer/Fillet** from the adaptive menu, select an empty area of the grid to complete the tool.

Fillet types

You can create the following fillet types in Shapr3D:

- **Radius** – Rounds the edge by defining the curvature with a specific radius. This type of fillet is ideal when you want to ensure a consistent, circular transition along the edge.
- **Chordal** – Chord width, or chord, is the straight-line distance between two points on a curve, such as a circle or arc. A chordal fillet specifies the distance between the endpoints of the fillet, rather than the radius. This gives you greater control over how the edge transitions, especially in cases where the exact distance between endpoints is critical.

Understanding fillet settings

Fillets can be modified to adjust their profile, corner type, and continuity conditions. You can use the on-screen elements shown below to make these adjustments:

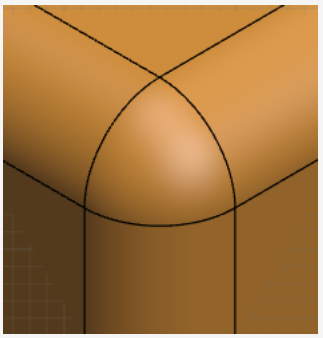
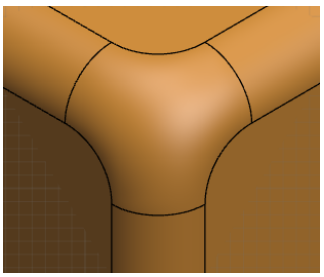
Radius dimension label

This dimension label defines the radius for a default circular fillet. If you make changes to the profile or continuity type, this value represents the setback distance from the original edge, rather than a true radius value.

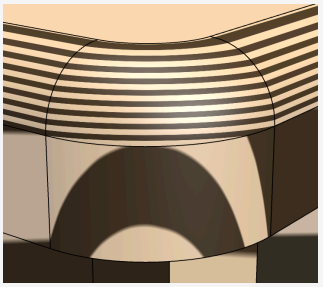
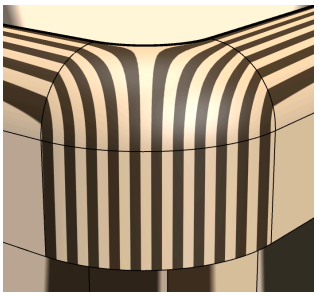
Settings badge

Select the Settings badge to access the menu with the **Corners** and **Continuity** options.

Corners

Rolling Ball	This type of corner has a constant radius. This is the default corner type.	
Setback	This type of corner sets back the edges where the fillets begin blending into the corner. The blended face has a variable radius.	

Continuity

G1	Use this continuity type to maintain tangency between the fillet and adjacent faces. This is the default continuity type.	
G2	Use this continuity type to create a fillet that matches the curvature of adjacent faces. This produces a smoother transition between the fillet and adjacent faces than G1 continuity. Fillets with G2 continuity will have a non-circular cross-sectional profile.	

> ### Tip
>
> Zebra stripes are shown in the images above to help visualize curvature conditions. For more information on this tool, see Views and Appearance.

Profile slider

The profile slider can change the cross-sectional profile of the fillet. Sliding toward the right will produce a sharper profile and sliding to the left will produce a flatter profile.

Note: Setting a different value than the default will result in losing the radial dimension of the fillet.

Magnitude dimension label

You can use this dimension label to define the magnitude of change for the profile slider. The value can be between -1 and 1. A value of -1 would be a completely flat face and positions the slider all the way to the left. A value of 1 would result in a sharp corner and positions the slider all the way to the right.

History settings

Chamfer

To manage history settings for the Chamfer action, find the step for your specific chamfer and then view or modify the following:

- **Edge** – Click/tap **Edit…** or **Select…** to choose a different edge to chamfer and then select **Done** to finish.
- **Distance #1** – Enter your preferred first distance for a 2-distance chamfer.
- **Distance #2** – Enter your preferred second distance for a 2-distance chamfer.
- **Include Tangent Edges** – Turn on to include tangent edges for the chamfer.

Fillet

To manage history settings for the Fillet action, find the step for your specific fillet and then view or modify the following:

- **Edge** – Click/tap **Edit…** or **Select…** to choose a different edge to fillet and then select **Done** to finish.
- **Size Type** – Select the dropdown to choose between a **Radius** or **Chord Width** fillet.
- **Radius** – For radius fillets, enter your preferred radius.
- **Width** – For chordal fillets, enter your preferred width.
- **Continuity** – Select the dropdown to choose between a **G**1 and **G2** continuity.
- **Corner** – Select the dropdown to choose between a **Rolling Ball** or **Setback** corner.
- **Overflow** – Select the dropdown to choose between an **Auto**, **Cliff**, **Smooth**, and **Notch** overflow**.**
- **Curvature** – Enter your preferred curvature value for the fillet.
- **Include Tangent Edges** – Turn on to include tangent edges for the chamfer.
- **Y-Shaped Blend** – Toggle the slider to turn on or turn off Y-shaped blend.

Video walkthrough

Shapr3D Manual - Chamfer/Fillet | Tools

Extrude

You can use the **Extrude** tool to create 3D geometry by pushing or pulling a face or closed sketch in a linear direction. If necessary, you can specify a draft angle, which adds a taper to the side faces of the geometry.

The type of extrude created is automated based on geometric conditions but can be modified if necessary. Using the **Extrude** tool can result in geometry that is:

- A new body.
- Added to an existing body.
- Removed from an existing body, creating a hole or void.
- The intersection of the revolved profile with an existing body.

How to use the Extrude tool

To use the **Extrude** tool:

1. Go to **Tools** > **Extrude** tool.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the face(s) or sketch profile(s) you want to extrude.
3. Use the gizmo to extrude the selected items to your desired distance.
4. **Optional**: Use the Boolean badge to manually select a boolean action.
5. **Optional**: Use the gizmo to define a draft angle.
6. Select **Done**.
   Note: When using **Extrude** from the adaptive menu, select an empty area of the grid to complete the tool.

History settings

To manage history settings for the Extrude action, find the step for your specific extrusion and then view or modify the following:

- **Profile** – Click/tap **Edit…** or **Select…** to choose a different sketch profile for the Extrude action.
- **Sides** – Select the dropdown to choose between **One-Sided** and **Symmetric** for your extrude extension.
- **Extent** – Select the dropdown to choose between **Distance**, **To Object**, and **Through All** as the extent type for the extrusion.
    - **Distance** – Enter your preferred distance.
    - **To Object** – Click/tap **Select…** beside **To Object** to select the object where you want to extend the extrusion.
    - **Through All** – Choose whether to turn on or turn off **Flip Direction**.
- **Draft Angle** – Enter your preferred draft angle for the extrusion.
- **Start** – To choose from where you want to start your extrusion, select the dropdown to choose between **From Profile**, **Offset**, and **From Plane**.
    - **From Profile** – Starts your extrusion from the selected profile.
    - **Offset** – Enter values for the **Start Offset** and **End Offset** fields.
    - **From Plane** – Select **Edit…** beside **Start Plane** to choose from which plane you want the extrusion to start.
- **Result** – Select the dropdown to choose between **New Body**, **Union**, **Intersect**, and **Subtract** as your preferred extrude type and select **Selec…** beside **Target** to select the target bodies for your extrude type.

Automatic Boolean actions

The type of Boolean operation created is automated based on geometric conditions. Here is a summary of the automatic Boolean actions and when they are triggered:

Boolean type	Result	Automatic trigger
New Body 	Creates a new standalone body.	Automatically triggers when new geometry does not intersect or make contact with any existing body.
Union 	Executes a **Union** boolean operation, adding the new geometry directly to an existing body and sharing intersecting edges.	Triggers with any of the following conditions: • A face on an existing body is selected to generate the geometry path. • A sketch profile connected to an existing body is pulled or extended away from it.
Subtract 	Executes a **Subtract** boolean operation, removing the new geometry from an existing body to produce a cut, hole, or void.	Automatically triggers when a sketch profile or face is selected and pushed or extended directly into an existing solid body.
Intersect 	Restricts the output to an **Intersect** boolean type, retaining only the shared volume where the new geometry overlaps an existing body.	An intersection is not applied automatically and must be selected manually from the Boolean badge menu.

Extrude vs Offset Face

While the tools look similar, **Offset Face** cannot be used to create a body from a closed sketch, while **Extrude** created a 3D body from a 2D sketch by linearly extending a face on a body.

- **Offset Face**: Adjacent faces are extended or trimmed.
- **Extrude**: Creates a new prismatic shape from a face, while offset modifies a body.

Additionally, you can only **Extrude** planar faces, but this is not a limitation of the **Offset Face** tool.

Video walkthrough

Shapr3D Manual - Extrude | Tools

Shell

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You can use the **Shell** tool to create models with a uniform wall thickness. This tool allows you to convert a face of any solid geometry into a hollow shell with a defined wall thickness.

How to use the Shell tool

To use the **Shell** tool:

1. Go to **Tool** > **Shell**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the face or faces to remove.
3. Use the gizmo to define the wall thickness for the shell.
4. Select **Done**.
   Note: When using **Shell** from the adaptive menu, select an empty area of the grid to complete the tool.

History settings

To manage history settings for the Shell action, find the step for your specific shell and then view or modify the following:

- **Target** – Click/tap **Edit…** or **Select…** to choose a different target for the face you want to shell and then select **Done** to finish.
- **Thickness** – Enter your preferred shell thickness.

Video walkthrough

Shapr3D Manual - Shell | Tools

Loft

You can use the **Loft** tool to create complex shapes defined by cross-sectional profiles. This tool interpolates the shape of the body between the cross-sections you specify. For more control over the shape in between the profiles, you can create guide curves and make use of connection points.

How to use the Loft tool

Follow these steps to use the **Loft** tool.

1. Create the necessary loft elements:
    - For the loft profiles, draw closed sketches on separate planes.
    - Optionally, sketch guide curves that intersect every cross-section.
2. Go to **Tools** > **Loft**.
   For other methods of accessing this tool, visit Accessing tools.
3. Select the loft profiles. If more than two profiles are being used, select them in the order they should be connected.
4. If necessary, adjust the loft connection point and select the loft guide curves.
5. **Optional**: For lofts with periodicity, you can blend or smoothen the first and last profile by turning on **Periodic Loft** in the History sidebar.
6. Select **Done**.
   Note: When using **Loft** from the adaptive menu, select an empty area of the grid to complete the tool.

Understanding Loft elements

Elements of a loft include cross-sectional profiles, optional guide curves, and control points. Learn about the different loft elements below.

Loft profiles

Profiles can be faces or sketch profiles that represent cross-sections of the completed shape. There must be a minimum of two loft profiles — one located at the beginning of the shape, and one at the end of the shape. Additional profiles can be defined between the start and end profiles as necessary.

Note: Loft profiles can only be single closed cross-sections. Two cross sections from the same plane or intersecting cross sections cannot be used.

Connection Points

Connection points are automatically generated between loft profile vertices. They help identify how the vertices from one profile are mapped to the vertices in the next profile. Connection points can be dragged to modify how the vertices are mapped. Making changes to connection points will often introduce twist into the loft feature, or can remove twist. Connection points will not be available if guide curves are specified for a loft.

Guide Curves

Guide curves are an optional loft element. They can be used to control the shape of the loft in between the loft profiles. You can use sketch curves or edges as loft guide curves.

Note: Make sure to connect the guides to the cross sections. If the guides are not intersecting with the cross section curve or they are inside the cross section, the loft will fail.

History settings

To manage history settings for the Loft action, find the step for your specific loft and then view or modify the following:

- **Profiles** – Click or tap **Edit...** or **Select**... to choose different loft profiles.
- **Periodic Loft** – For curved lofts, turn on to blend or smoothen the first and last profile.
- **Guides** – Click or tap **Edit...** or **Select**... to choose different loft guides.
- **Start Tangent Continuity** – Select the dropdown to choose between a **None**, **G1**, and **G2** start tangent continuity.
- **Start Tangent Magnitude** – Enter a value for the start tangent magnitude.
- **End Tangent Continuity** – Select the dropdown to choose between a **None**, **G1**, and **G2** end tangent continuity.
- **End Tangent Magnitude** – Enter a value for the end tangent magnitude.

Video walkthrough

Shapr3D Manual - Loft | Tools

Union

In CAD modeling, you can use boolean operations to create complex shapes and forms of your models.

The **Union** tool is one of the boolean operations in Shapr3D that you can use to merge separate bodies into a single, united body. Bodies must be overlapping to be valid for this tool.

How to use the Union tool

To use the **Union** tool:

1. Go to **Tools** > **Union**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the bodies you want to merge.
   Note: In History, the last body you select becomes your default **Target** parameter while the remaining bodies will be your default **Tool** parameters.
3. If necessary, adjust the **Keep Originals** setting with any of the following options:
   - **Keep Originals: Off:** The default **Off** setting for **Keep Originals** results in the original bodies being absorbed into the merged body.
   - Keep **Originals: On:** By turning **On** the setting for **Keep Originals**, the results of the Union will be created as a new merged body, and the original separate bodies will still be available within the project.
4. Select **Done**.

History settings

To manage history settings for the Union action, find the step for your specific union and then view or modify the following:

- **Target** – Click/tap **Edit…** or **Select…** to choose a different target for the bodies you want to union and then select **Done** to finish.
- **Tool** – Click/tap **Edit…** or **Select…** to choose a different tool for the bodies you want to union and then select **Done** to finish.
- **Type** – Select the dropdown to update the boolean type to either **Union**, **Subtract**, or **Intersect**.
  Note: Make sure to update the **Target** and **Tool** settings that apply to your selected boolean type.
- **Keep Target Bodies** – Turn on to keep target bodies.
- **Keep Tool Bodies** – Turn on to keep tool bodies.

Video walkthrough

Shapr3D Manual - Union | Tools

Subtract

In CAD modeling, you can use boolean operations to create complex shapes and forms of your models.

Subtract is one of the boolean operations in Shapr3D that you can use on overlapping bodies to remove or subtract the volume of a selected body from another body. To use this tool, you must have at least two overlapping bodies.

How to use the Subtract tool

To use the **Subtract** tool:

1. Go to **Tools** > **Subtract**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the target body or bodies that you want to remove from. Target bodies are highlighted in purple and marked with a target badge (+).
   Note: In History, this body will be your **Target** parameter. You can select the badge to toggle the desired state of the selected body.
3. Select the bodies that you want to subtract from the target body. These bodies are highlighted in blue and marked with a subtract badge (-).
   Note: In History, this body will be your **Tool** parameter. You can select the badge to toggle the desired state of the selected body.
4. If necessary, adjust the **Keep Originals** settings.
5. To finish, select **Done**.

Subtract tool settings

When using the Subtract tool, you can adjust the Keep Original settings to control whether the body being modified, the body being removed, or both are retained after the operation. These settings are also available in the History settings after completing the operation.

You can keep the original of the following:

- **All** - With this option selected, the results of the **Subtract** tool are created as a new body, and the original bodies will still be available.
- **Modified Bodies** - The bodies being modified are the target bodies (highlighted in purple). With this option selected, the results of the Subtract are created as a new body and the original target bodies will still be available.
- **Removed Bodies** - The bodies being removed. With this option selected, the results of the **Subtract** tool are created as a new body and the original bodies will still be available. The target bodies will be removed.
- **None** - With this option selected, only the results of the **Subtract** tool remain. The original bodies will be removed.

History settings

To manage history settings for the Subtract action, find the step for your specific subtraction and then view or modify the following:

- **Target** – Click/tap **Edit…** or **Select…** to choose a different body you want to subtract from and then select **Done** to finish.
- **Tool** – Click/tap **Edit…** or **Select…** to choose a different body you want to subtract and then select **Done** to finish.
- **Type** – Select the dropdown to update the boolean type to either **Union**, **Subtract**, or **Intersect**. **Note**: Make sure to update the **Target** and **Tool** settings that apply to your selected boolean type.
- **Keep Target Bodies** – Turn on to keep target bodies.
- **Keep Tool Bodies** – Turn on to keep tool bodies.

Video walkthrough

Shapr3D Manual - Subtract | Tools

Intersect

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In CAD modeling, you can use boolean operations to create complex shapes and forms of your models.

Intersect is one of the boolean operations in Shapr3D that you can use to create new bodies from the intersection of overlapping bodies. To use this tool, you must have at least two overlapping bodies.

How to use the Intersect tool

To use the **Intersect** tool:

1. Go to **Tools** > **Intersect**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the target body or bodies that you want to modify. Target bodies will be highlighted in purple and marked with a target badge.
   Note: In History, this body will be your **Target** parameter. You can select the badge to toggle the desired state of the selected body.
3. Select the intersecting body or bodies. These bodies are highlighted in blue and marked with a tool badge.
   Note: In History, this body will be your **Tool** parameter. You can select the badge to toggle the desired state of the selected body.
4. **Optional**: Adjust the **Keep Originals** settings.
5. To finish, select **Done**.

Intersect tool settings

When using the **Intersect** tool, you can adjust the **Keep Originals** settings to control whether the original bodies are retained after the operation. These settings are also available in the History settings after completing the operation.

- **Keep Originals: Off** - When set to Off, only the body resulting from the Intersect remains. The original bodies will be removed. This is the default setting.
- **Keep Originals: On** - When set to On, the results of the Intersect are created as a new body and the original bodies will still be available.

History settings

To manage history settings for the Intersect action, find the step for your specific intersection and then view or modify the following:

- **Target** – Click/tap **Edit…** or **Select…** to choose a different body you want to intersect and then select **Done** to finish.
- **Tool** – Click/tap **Edit…** or **Select…** to choose a different body you want to modify/intersect with and then select **Done** to finish.
- **Type** – Select the dropdown to update the boolean type to either **Union**, **Subtract**, or **Intersect**. **Note**: Make sure to update the **Target** and **Tool** settings that apply to your selected boolean type.
- **Keep Target Bodies** – Turn on to keep target bodies.
- **Keep Tool Bodies** – Turn on to keep tool bodies.

Video walkthrough

Shapr3D Manual - Intersect | Tools

Split Body

You can use the Split Body tool to divide a solid body into multiple parts without removing material. Create precise cuts with construction planes, grid planes, sketch profiles, faces, imported images, or even one of the body's own faces to separate it into distinct solids.

How to split bodies

To use the **Split Body** tool:

1. Go to **Tools** > **Split Body.**
   For other methods of accessing this tool, visit Accessing tools.
2. Select the bodies that you want to split. Bodies to split will be highlighted in blue and marked with the Split Body badge.
3. Select a construction plane, grid plane, sketch profile, face, or image for the splitting tool.
   Note: Select the badges in the workspace to redefine or toggle selections.
4. **Optional**: Adjust the **Keep Originals** settings.
5. To finish, select **Done**.

Split Body settings

When using the **Split Body** tool, you can adjust the **Keep Originals** settings to control whether the original body is retained after the operation. These settings are also available in the History settings after completing the operation.

- **Keep Originals: Off** – When set to **Off**, only the bodies resulting from the Split Body tool will remain. The original body will be removed. This is the default setting.
- **Keep Originals: On** – When set to **On**, the results of the Split Body tool will be created as new bodies and the original body will still be available.

Splitting tools

Splitting tools define how and where a body is divided. When you select a splitting tool, it's projected through the selected bodies, so it doesn't need to be in direct contact. You can select multiple splitting tools at once to create several cuts in a single operation.

You can use any of the following as splitting tools:

- **Construction or grid planes** – Ideal for precise, planar cuts across a body.
- **Sketch profiles** – Useful for custom or curved cuts. Multiple profiles in the same plane merge into one splitting surface.
- **Body faces and edges** – You can use the faces or coplanar edges of another body as a splitting surface.
  Note: If the splitting sketches are not connected, they will not merge and will instead create separate simultaneous cuts.
- **Imported images** – Reference an image as a cutting guide.
- **Body's own face** – Split a body using one of its own faces to create separate solids.

History settings

To manage history settings for the Split Body action, find the step for your specific split body and then view or modify the following:

- **Bodies to Split** – Click/tap **Edit…** or **Select…** to choose a different body you want to split and then select **Done** to finish.
- **Split With** – Click/tap **Edit…** or **Select…** to choose a different sketch, face, plane, edge, or plane you want to use as the splitting tool and then select **Done** to finish.
- **Keep Originals** – Toggle the slider to turn on or off the original body.

Video walkthrough

Shapr3D Manual - Split Body | Tools

Revolve

You can use the **Revolve** tool to create solid bodies that are symmetrical about an axis. Some shapes that are commonly created using **Revolve** include cylinders, cones, spheres, or any axial symmetry, round shapes. The **Revolve** tool also includes a height parameter that allows you to generate shapes such as coils, springs, and threads.

The type of revolve created is automated based on geometric conditions but can be modified if necessary. Using the **Revolve** tool can result in geometry that is:

- A new body.
- Added to an existing body.
- Removed from an existing body, creating a hole or void.
- The intersection of the revolved profile with an existing body.

How to use the Revolve tool

Create a basic revolve

Use the Revolve tool to rotate a profile around an axis and generate a solid body.

To create a basic revolve:

1. Go to **Tools** > **Revolve**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the profile and then select the axis.
3. **Optional**: Use the gizmo to adjust the angle for the revolve action.
4. **Optional**: Use the Boolean badge to manually select a boolean action.
5. **Optional**: To create a helical shape, specify a height for the revolve.
6. To finish, Select Done.
   Note: When using **Revolve** from the adaptive menu, select an empty area of the grid to complete the tool.

Create helical bodies

Add a height value to your revolve to generate coils, springs, or threaded shapes.

To create a helical body:

1. Go to **Tools** > **Revolve**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the profile and then select the axis.
3. Use the gizmo to adjust the angle for the revolve action.
   Note: Every 360° equals a complete revolution of the helix. You can use the calculator functionality to multiply 360 by the number of revolutions.
4. **Optional**: Use the Boolean badge to manually select a boolean action.
5. Use the gizmo to define the height of the helical body.
6. To finish, select **Done**.
   Note: When using **Revolve** from the adaptive menu, select an empty area of the grid to complete the tool.

Revolve profiles and axes

Creating a revolve requires a profile and an axis. The profile is the shape being rotated, and the axis is the line it rotates around. If a height is defined, the profile spirals upward as it rotates, which is how shapes like coils, springs, and threads are created. These settings are also available in the History settings after completing the operation.

- **Profile** – A closed sketch or a face to revolve around the axis.
- **Axis of revolution** – An axis of the grid, a construction axis, a sketch line, or a linear edge.
- **Angle** – By default, the profile is revolved by 360°, but you can modify the angle as needed.
- **Height** – To generate a helical shape, you can specify a height for the revolve action.

Automatic Boolean actions

The type of Boolean operation created is automated based on geometric conditions. Here is a summary of the automatic Boolean actions and when they are triggered:

Boolean type	Result	Automatic trigger
New Body 	Creates a new standalone body.	Automatically triggers when new geometry does not intersect or make contact with any existing body.
Union 	Executes a **Union** boolean operation, adding the new geometry directly to an existing body and sharing intersecting edges.	Triggers with any of the following conditions: • A face on an existing body is selected to generate the geometry path. • A sketch profile connected to an existing body is pulled or extended away from it.
Subtract 	Executes a **Subtract** boolean operation, removing the new geometry from an existing body to produce a cut, hole, or void.	Automatically triggers when a sketch profile or face is selected and pushed or extended directly into an existing solid body.
Intersect 	Restricts the output to an **Intersect** boolean type, retaining only the shared volume where the new geometry overlaps an existing body.	An intersection is not applied automatically and must be selected manually from the Boolean badge menu.

History settings

To manage history settings for the Revolve action, find the step for your specific revolution and then view or modify the following:

- **Profile** – Click/tap **Edit…** or **Select…** to choose a different sketch profile for the Revolve action and then select **Done** to finish.
- **Axis** – Click/tap **Edit…** or **Select…** to choose a different reference axis for the Revolve action and then select **Done** to finish.
- **Angle** – Enter your preferred angle.
- **Elevation** – Enter your preferred elevation distance.

Video walkthrough

Shapr3D Manual - Revolve | Tools

Sweep

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The **Sweep** tool a 3D body by extending a profile along a path. The profile can be a closed sketch or face, and the path can be sketch elements or edges of a body.

Use Sweep when you need geometry that follows a defined route, such as:

- Pipes, tubes, and hose-like parts
- Wires, cables, and routed channels
- Handles, rails, and frame-like shapes
- Profiles that need to follow curved or angled edges

Note: For shapes that transition between different profiles, use the **Loft** tool.

How to use the Sweep tool

To use the **Sweep** tool:

1. Go to **Tools** > **Sweep** tool.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the profiles, paths, or spines for your sweep action.
3. To finish, select **Done**.

> **Tip**
>
> When using Sweep from the adaptive menu, select the profile and all spines before activating the tool.

Sweep profiles and paths

To create a Sweep, you need a profile and a path or spine. The profile is the shape you want to move, and the path is the line it follows.

- **Profile** – A sketch region or face that acts as the cross-section for the new body. For the best results, ensure your profile is positioned at the beginning of your path.
- **Path** – Sketch elements or edges (also known as a spine) that the profile will follow. If your path has sharp corners, try adding fillets to the sketch; this prevents the profile from "self-intersecting" or failing during the operation.

History settings

- **Profile** – Click/tap **Edit…** or **Select…** to choose a different sketch profile or face, then select **Done** to finish.
- **Path** – Click/tap **Edit…** or **Select…** to choose a different path, then select **Done** to finish.
- **Profile Position** – Controls how the profile is positioned relative to the path:
  - **Auto** – Uses the default alignment based on your selection.
  - **Path intersection** – Positions the profile where it intersects the path.
  - **Closest point** – Positions the profile at the closest point on the path.

    - **Closest endpoint** – Positions the profile using the nearest path endpoint.
- **Orientation** – Controls how the profile is oriented as it follows the path:
    - **Normal to path** – Keeps the profile perpendicular to the path direction.
    - **Parallel to profile** – Keeps the profile aligned to its original orientation.
- **Twist** – Rotates the profile along the path by the specified angle.
- **Scale** – Adjusts the size of the profile along the path.
- **Corner type** – Controls how corners are handled along the sweep path:
    - **Mitre** – Keeps corners sharp.
    - **Round** – Rounds corners for smoother transitions.

Video walkthrough

Shapr3D Manual - Sweep | Tools

Replace Face

You can use the **Replace Face** tool to extend or trim a selected face to match another face. **Replace Face** can be a useful tool to modify flat planar faces generated by tools such as extrude. It can also be an effective method for matching the faces of connected bodies.

How to use the Replace Face tool

To use the **Replace Face** tool:

1. Go to **Tools** > **Replace Face**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the face or faces you want to replace. Faces to replace will be highlighted in blue and marked with the Replace Face badge.
   Note: You can select the badge to redefine a face as a face to be replaced or a target face.
3. Select the replacement face. This face will be highlighted in purple and marked with the target badge. The face you are replacing will be extended or trimmed to meet the target face.
   Note: You can select the badge to redefine a face as a face to be replaced or a target face.
4. Select **Done.**

History settings

To manage history settings for the Replace Face action, find the step for your specific replacement face and then view or modify the following:

- **Faces to Replace** – Click/tap **Edit…** or **Select…** to choose a different face to replace and then select **Done** to finish.
- **Replacing Face** – Click/tap **Edit…** or **Select…** to choose a different replacing face and then select **Done** to finish.
- **Flip Alignment** – Turn on to extend the face replacement to the other side of the replacing face.

Video walkthrough

Shapr3D Manual - Replace Face | Tools

Offset Edge (3D)

The **Offset Edge** tool creates new sketch elements that are offset from existing elements by a specified distance. You can use the **Offset Edge** tool within a sketch, with sketch profiles, or with the edges and faces of a 3D body.

How to use the Offset Edge tool

To offset from a 3D body:

1. Go to **Tools** > **Offset Edge.**
   For other methods of accessing this tool, visit Accessing tools.
2. Select **Type** to access the **Offset Edge Type** menu.
3. Choose an offset type:
   - **Single** – Offsets a single edge from a 3D body.
   - **Chain** – Offsets a chain of edges from a 3D body.
4. Select the edge you want to offset.
5. Use the gizmo to specify the offset direction and distance.
6. To finish, select **Done**.

History settings

To manage history settings for the automatic Offset Edge action, find the step for your specific sketch and then view or modify the following:

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.

Video walkthrough

Shapr3D Manual - Offset Edge (3D) | Tools

Read more

For information on using Offset Edge within a sketch, visit Offset Edge (Sketch).

Project

You can use the **Project** tool to cast or project sketches, edges, or faces onto another face or sketch.

Projections create a relation or connection with the projected items that allows you to:

- Project different types of items onto complex, non-planar faces.
- Create sketches on faces based on existing 3D geometry.
- Create reference geometry that you can use for other operations.
- Merge sketches that are on the same plane but belong to separate sketches.

Create projections of items

To use the **Project** tool:

1. Go to **Tools** > **Project**.
   For other methods of accessing this tool, visit Accessing tools.
2. Choose a **Projection Type**.
   - **Edges** – The resulting projection will create new edges.
   - **Sketches** – The resulting projection will create new sketches.
     Note: You can also access this option while on sketch mode under **Sketch** > **Project - Sketches**.
3. Select the item you want to project. In History, this selection becomes your **Items to Project** parameter.
   Note: Each selection is indicated with a blue badge. You can click/tap the badge to change or cancel the selection.
4. Select the surface that you want to project on. In History, this selection becomes your **Projection surface** parameter.
   Note: The target surface will be colored purple, and you'll see a preview of your projection. You can click/tap the badge to change or cancel the selection.
5. To finish, select **Done**.

Edge projections

Unlike sketch projections, edge projections are created like geometrical properties of the body.

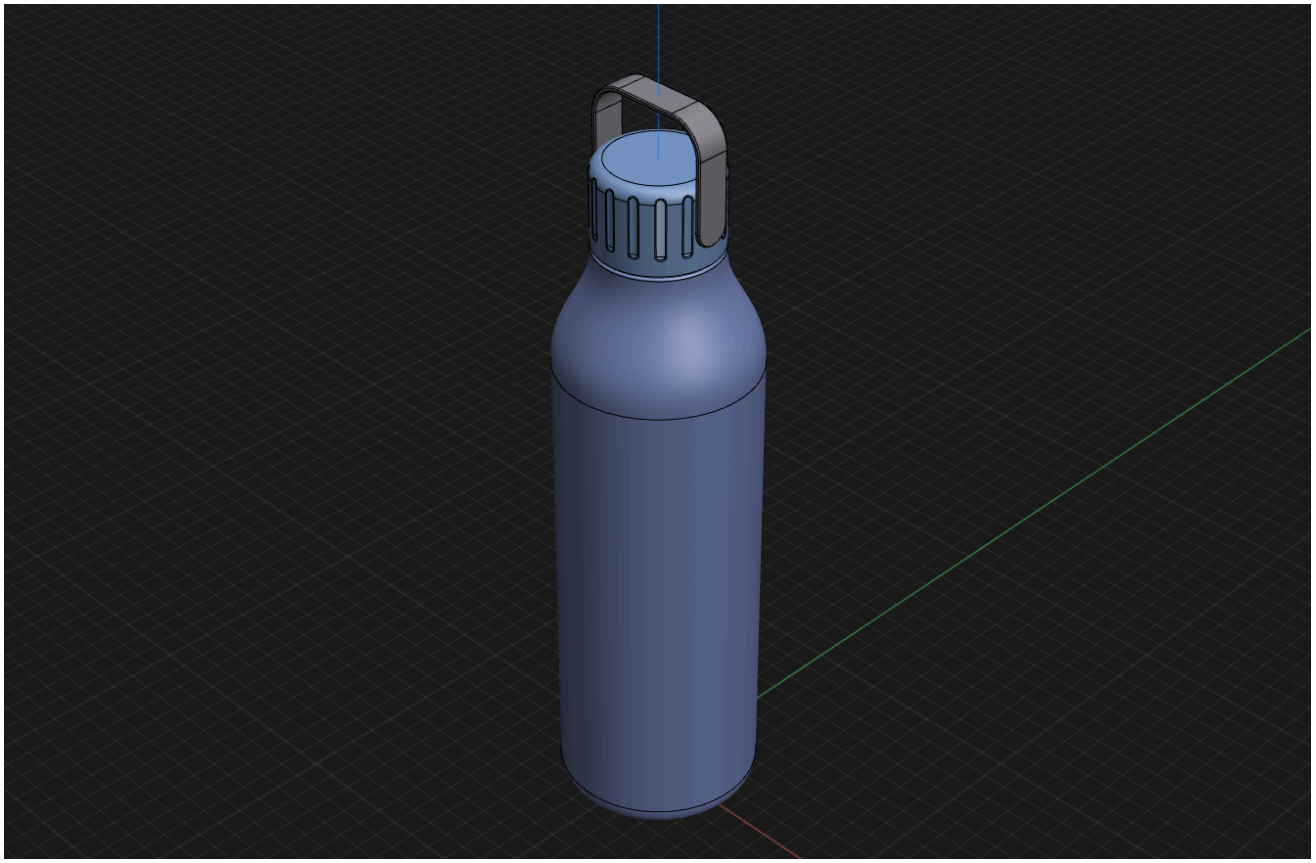
When you project your items as edges to a surface of a body, the resulting edge projection:

- splits the surface into different sections
- is always visible with the body even if sketches are hidden
- automatically follows any changes or adjustments you make to the shape or size of the original item you projected

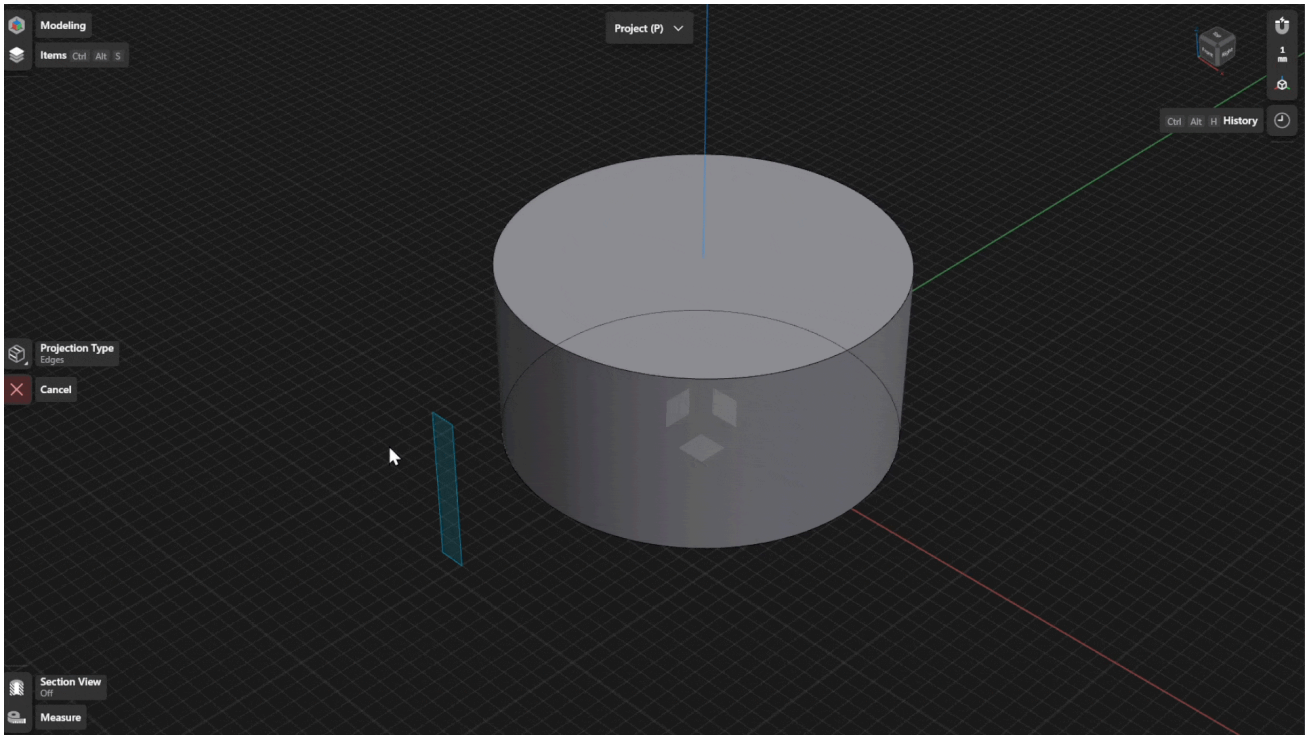
Check out some examples below to learn how you can use edge projections in your modeling workflow.

Example 1

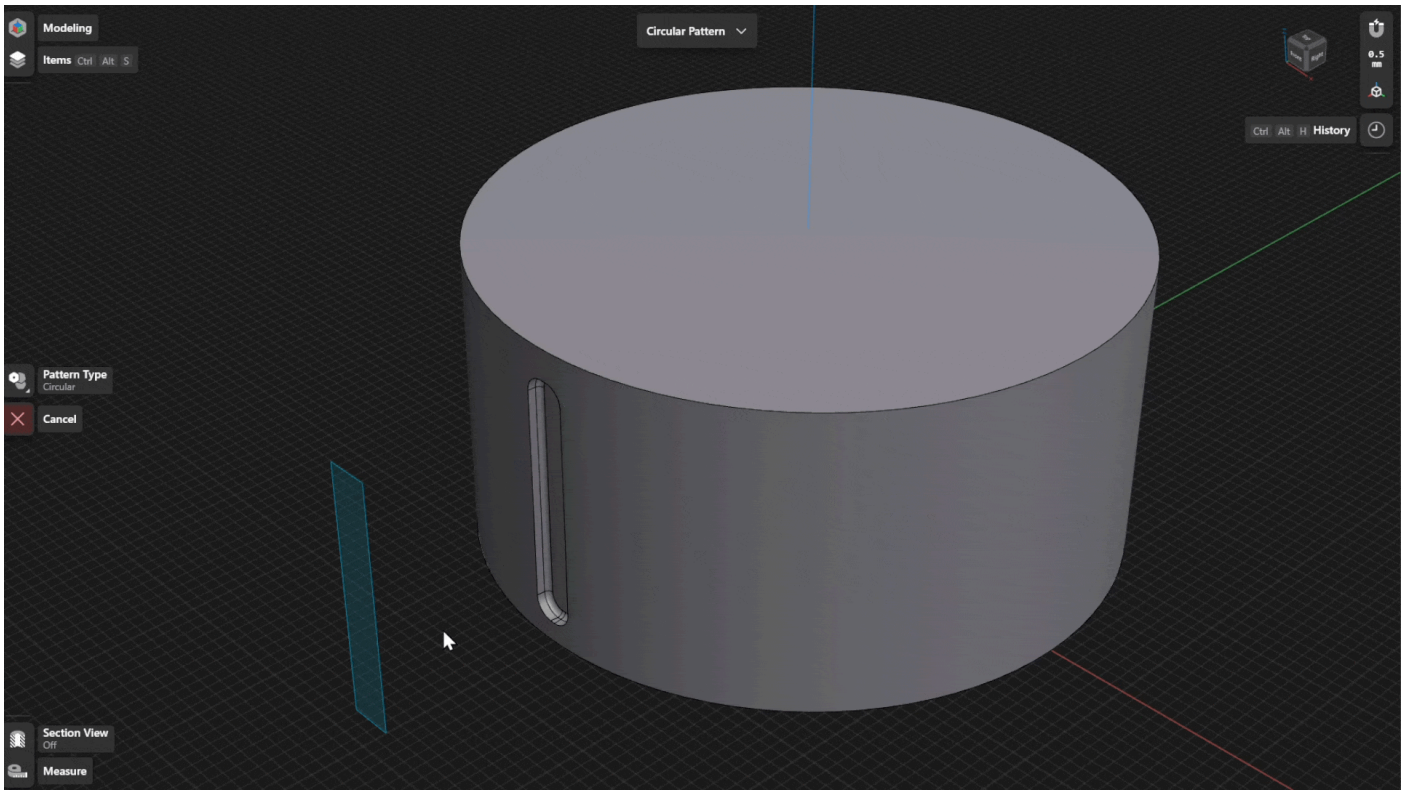
In the example model, we have a bottle with groove designs on the cap.



To achieve these grooves, we can use the **Project** tool to project a rectangular-shaped edge onto a cylindrical body and then use the **Chamfer/Fillet** tool to create the desired shape and curves:

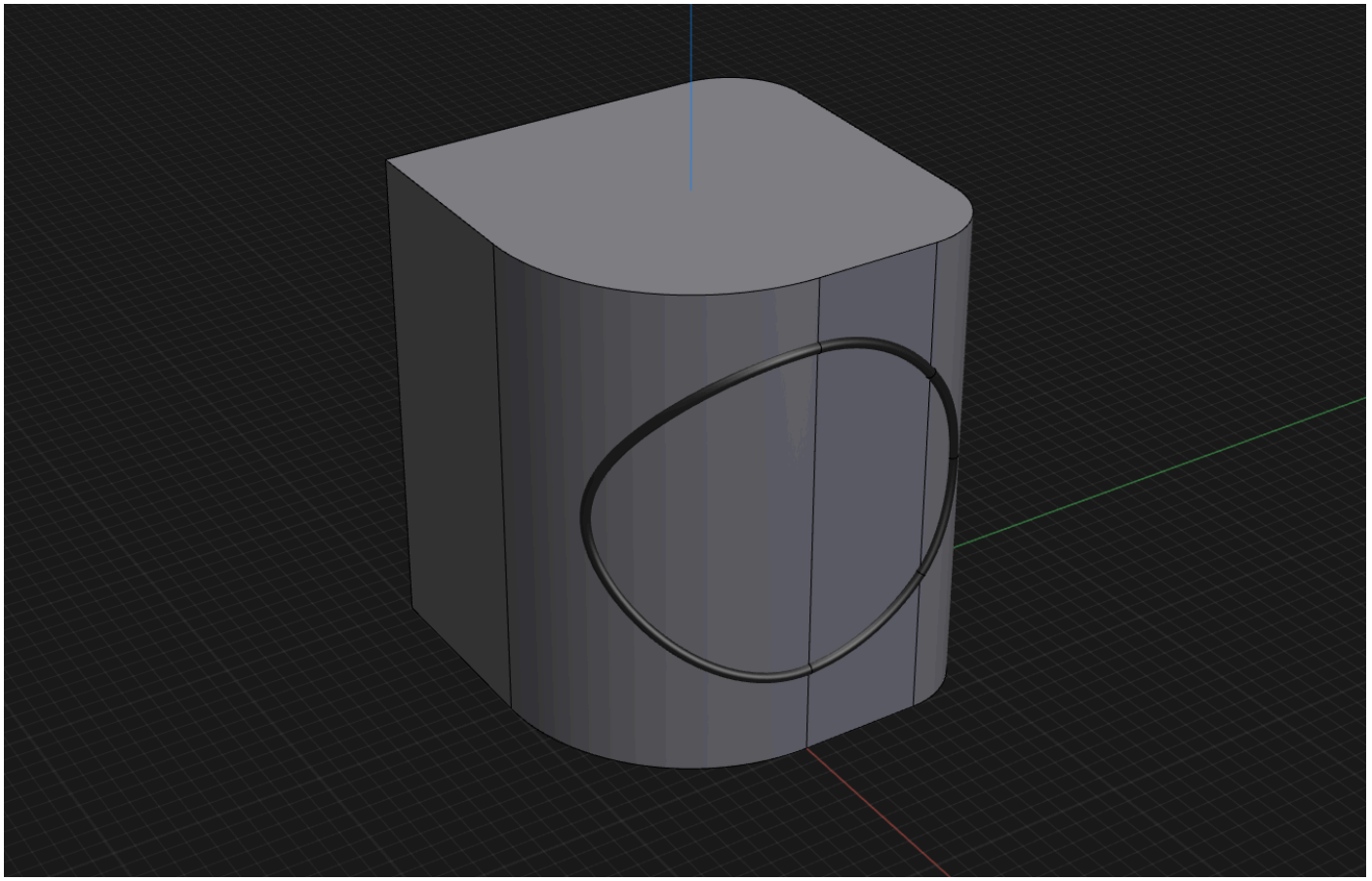


Lastly, you can use the **Pattern** tool to create a circular pattern around the cylinder.

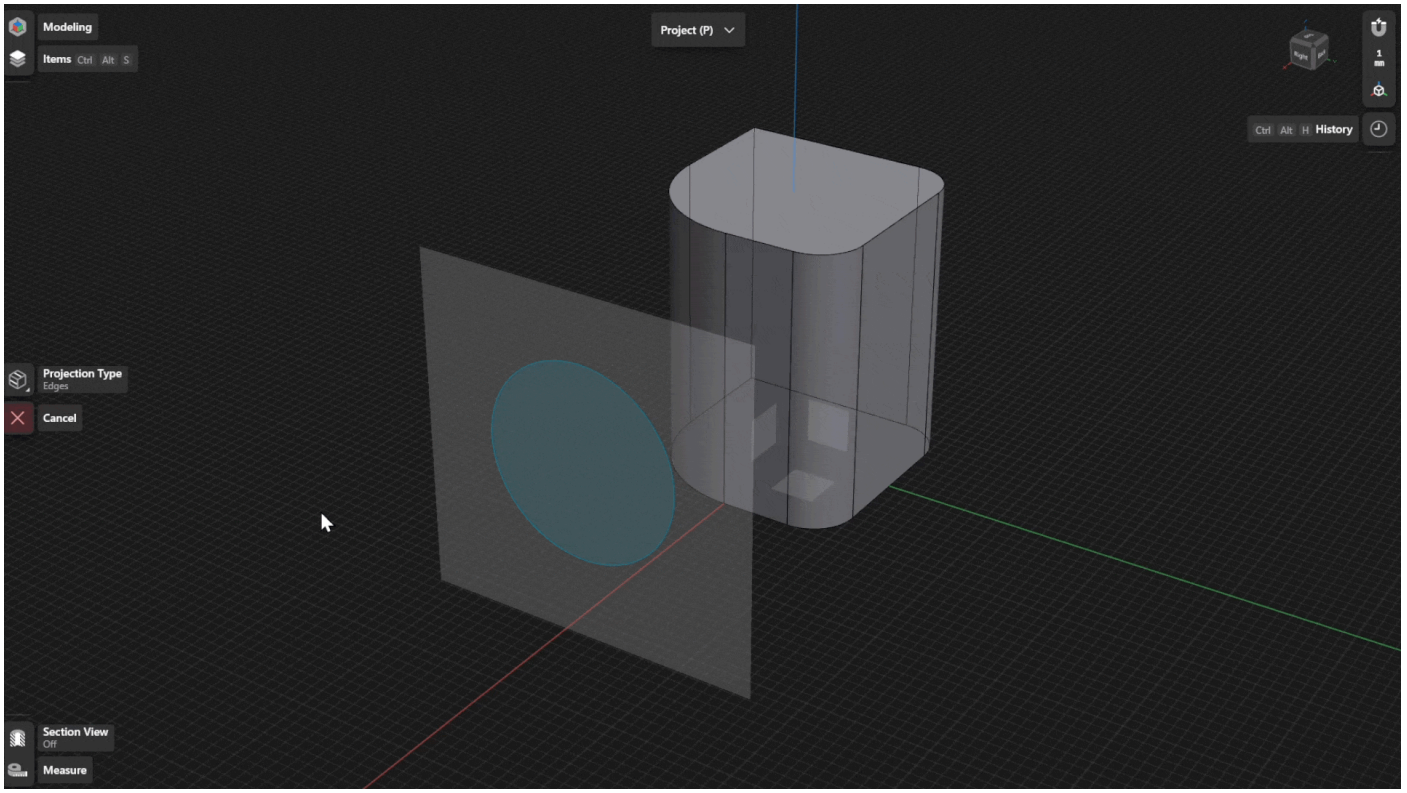


Example 2

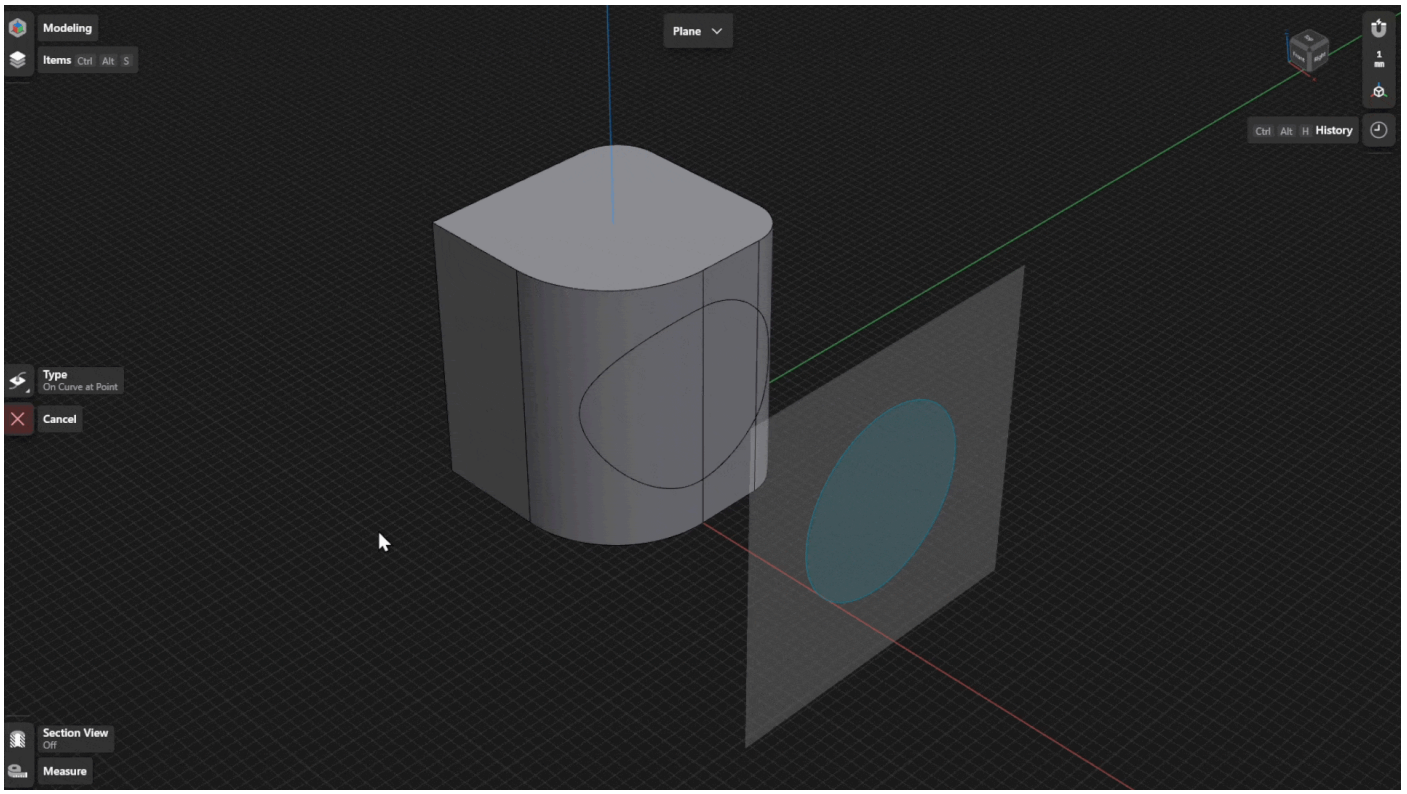
In the example model below, we have a circular ridge on a non-planar surface of the body.



To create this kind of ridge on this particular surface, we can use the **Project** tool to create a circular projection onto the applicable surfaces.



We can then use the **Sweep** tool to create the ridge by using the circular projection as the spine or path. To create the sketch profile, we can simply add an **On Curve at Point** construction plane to the circular projection.



Sketch projections

Sketch projections are useful for creating references or visualizing aspects of your design such as when you need a reference to better understand spatial relationships between different parts of your model. You can also use the **Project** tool to merge sketches that are on the same plane but exist in separate sketches, combining them into a single unified sketch for easier management and editing.

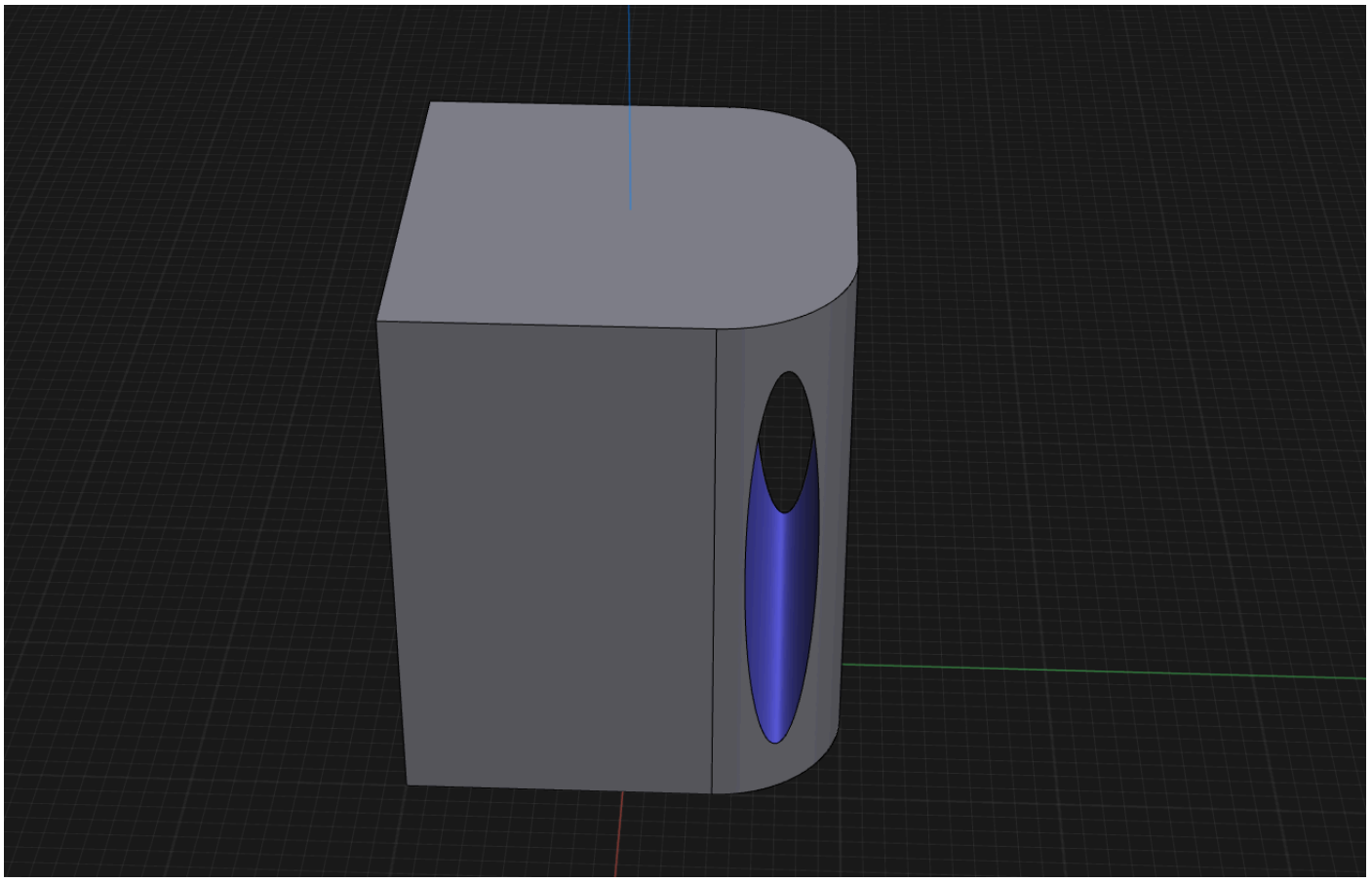
When you project your items as sketches to a surface of a body, the resulting sketch projection:

- is treated like a regular sketch item that can be hidden in the Items Manager
- can be used a reference sketches similar to how you treat sketches when directly sketching on a face

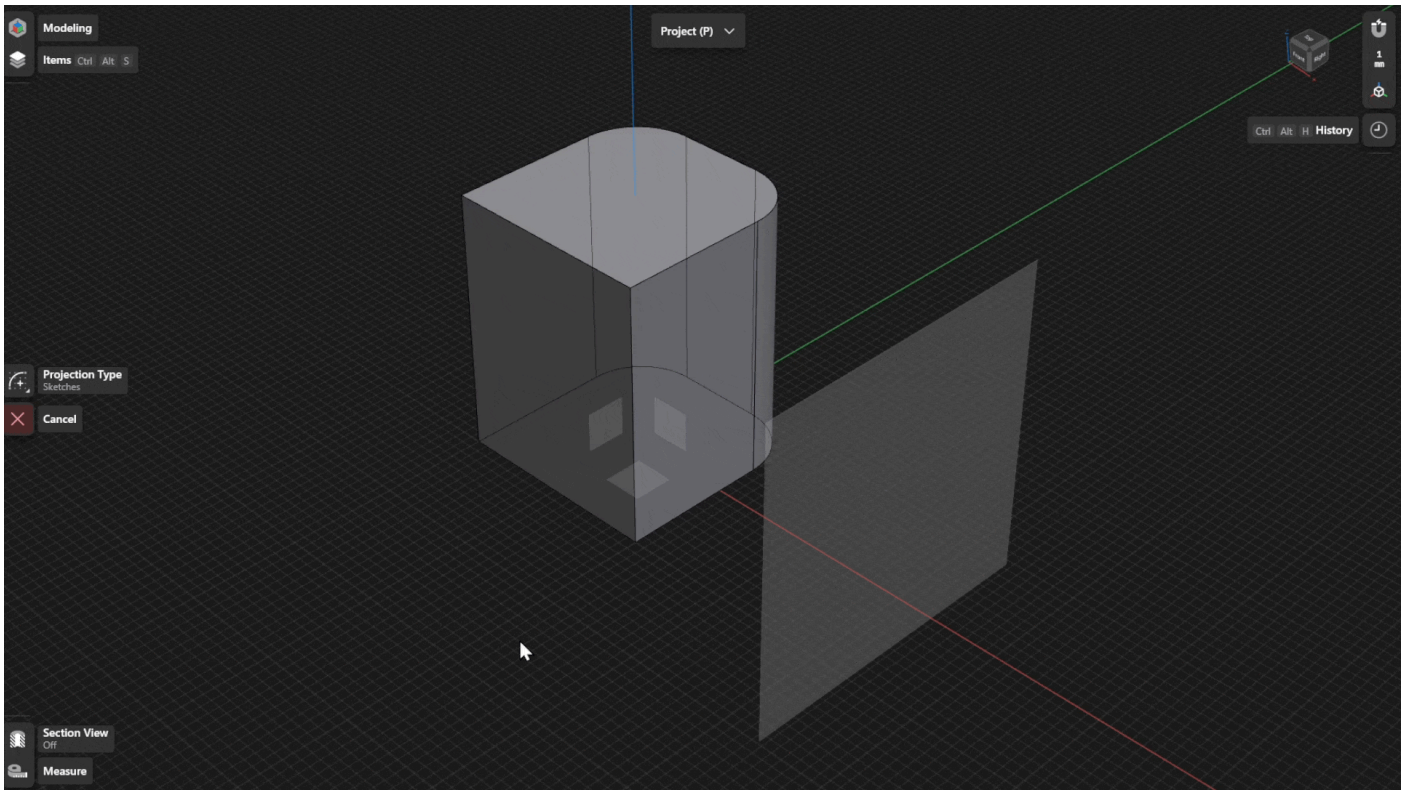
Check out the example below to learn how you can use sketch projections in your modeling workflow.

Example

In the example model below, we have an elliptical hole going through the rounded corners of the body.



To make a hole that perfectly fits the rounder corners, we can create a sketch projection of the curved edges onto a sketch plane.



We can then draw the ellipse on the sketch plane while using the sketch projections as guides and then use the **Extrude** tool to cut the hole shape we want.

Items to project

Items you can project onto another surface can be sketches, edges, faces, or entire bodies.

Items to project	Sample scenario
Sketches	You want to create the same sketch on a specific planar surface of your model.
Edges	You need a reference for the distance between edges when you need to create a sketch that fits the edge dimensions.
Faces	You want to create an engraving of a specific face onto a surface of your model.
Entire bodies	You want to create a 2D symbol for the shape of the entire body.

Projection surface

Surfaces you can project onto can be planar or non-planar surfaces:

- **Planar** – You can create both edge and sketch projections.
- **Non-planar** – You can only create edge projections. Sketch projections aren't possible similar to how you can't create sketches directly on non-planar surfaces.

Projection surface	Sample scenario
Planar	You want to project a complex shape onto a specific surface of your model.
Non-planar	You want a complex shape to conform to the curvature and contour of a specific surface of your model.

Relation between projections and projected items

When you create projections, a relation or connection is formed between the original item and the resulting projection.

Similar to the sketch pattern constraint concept, your projections will automatically follow any adjustments you make to the size or shape of the original item you projected.

History settings

To manage history settings for the Project action, find the step for your specific projection and then view or modify the following:

Edge projections

The default name for edge projections is **Projection**.

- **Projection Surface** – Click/tap **Edit…** or **Select…** to modify the projection surface and then select **Done** to finish.
- **Items to Project** – Click/tap **Edit…** or **Select…** to modify the items to project and then select **Done** to finish.

Sketch projections

Just like regular sketch items, the default name for sketch projections is **Sketch**.

- **Plane** – Click/tap **Edit…** or **Select…** to modify the plane or face where you want the sketch to be and then select **Done**.
- **Projection** – Click/tap **Edit…** or **Select**… to modify any existing projections for your sketch.

Video walkthrough

Shapr3D Manual - Project | Tools

Wrap & Emboss

You can use the **Wrap & Emboss** tool to map 2D sketch profiles onto cylindrical or conical faces. Unlike standard projection, this method maintains the surface-area dimensions of the original sketch, ensuring that geometry conforms to the curvature of the target body without distortion.

This feature can create a variety of functional and aesthetic elements, including:

- **Labels and Branding** – Apply precise logos, serial numbers, or text to curved packaging and mechanical components.
- **Functional Textures** – Create uniform knurling, ergonomic grips, or tread patterns on handles, dials, and wheels.
- **Structural Details** – Define recesses, grooves, or raised features that follow the exact taper of conical or cylindrical geometry.

How to use Wrap & Emboss

To use the **Wrap & Emboss** tool:

1. Go to **Tools > Wrap & Emboss**.
   For other methods of accessing this tool, visit Accessing tools.
2. Select the sketch profiles you want to use.
3. Select a single cylindrical or conical face as the target.
4. To finish, select **Done**.

Wrap & Emboss vs Project

Choosing between the **Wrap & Emboss** and **Project** tools depends on how you want the sketch geometry to interact with the target surface. While both tools move 2D sketches onto 3D bodies, they employ different geometric calculations that affect the final proportions of your design:

- **Wrap & Emboss** – Maps the sketch by "wrapping" it around the curvature. This method keeps the dimensions of your sketch the same and prevents the geometry from looking stretched.
- **Project** – Casts the sketch onto a face along a linear vector. On curved surfaces, this results in distortion as the sketch geometry is stretched to meet the face at varying angles.

History settings

To modify a **Wrap & Emboss** operation, find the step for your specific Wrap & Emboss action and then view or modify the following:

- **Items to Wrap** – The specific sketch profiles used in the operation.
- **Face to Wrap Onto** – The target cylindrical or conical surface.
- **Emboss** – The numerical value for the depth or height. Positive values create a raised surface; negative values create an engraving.
- **Rotation** – Rotates the sketch profiles around their own center point without changing their position on the face.
- **Center** – The origin point on the target surface used to align the sketch.

Video walkthrough

https://www.youtube.com/playlist?list=PLi4l3wxwkqywJ3jFxdT8b8O0473m0R3RT

Visualize

The **Visualize** tool from the **Tools** menu is another way to access the **Visualization** tool. To learn about this tool, go to Visualization.

Modes

Quick-toggle modes
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Isolate

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You can isolate items in your model to easily work on specific parts or bodies without distractions.

To isolate an item:

1. Select the item or items you want to isolate.
2. From the selection-based modes, select **Isolate**  to turn it on.
3. To turn off **Isolate**, select .

Section View

The **Section View** tool lets you look inside your 3D model by cutting through it along a selected plane or face. This is useful when analyzing internal details or checking cross-sections for fit and function.

When you turn on **Section View**, the visible part of your model is clipped based on the active plane, and cross-sections are rendered with clear fill patterns and colors to help you distinguish bodies more easily.

Create a section view

To create a section view in the modeling space:

1. Select a plane or planar face to define the clipping direction. or example, choose the **Top** plane to view a top-down cross-section.
2. From the modeling space modes, select  to turn on **Section view**.
   Note: You can also turn on Section View first and then select a plane to get the same result.
3. Once Section View is on and a plane is selected, you can:
   - Analyze the internal details and work directly with the geometry in the sectioned view.
   - Use the Section View options to control how the cross-section is displayed.
4. **Optional**: Adjust the cut using the gizmo on the plane:
   - Drag the arrows to move the plane deeper or shallower through the body.
   - Rotate the gizmo handles to tilt the cut at an angle.
   - Select the **Flip** badge to switch which side of the model is visible.
5. To turn off **Section View**, select .

You can also turn on **Section View** after selecting a plane or face to immediately generate a cross-section along that reference.

Section View visuals

When Section View is active, Shapr3D automatically applies section fills and contrasting colors to clipped bodies. This makes it easier to interpret cross-sections and inspect internal geometry.

- Cut areas are filled based on body properties for better contrast
- Intersections and overlaps between parts are easier to identify
- Useful for reviewing complex assemblies or multi-body designs

> **Tip**
>
> Section region colors follow your model's assigned body colors. If a body has a custom color, its section region will display the same color. Randomized section colors appear only when bodies use the default body color. If you notice unexpected colors, check your Visualization color properties to confirm the body color.

Section View options

Once Section View is active, use the following options to control how the cross-section is displayed:

- **2D Section View** – Displays the cross-section along with background edges and silhouettes for better depth perception while maintaining an orthographic perspective. If you rotate the camera away from the cut plane, select **Normal to Section** at the top of the workspace to instantly realign the view normal to the cut plane.
- **Section Only** – Limits the view to the cut area, hiding the rest of the model so you can focus on the cross-section.

> ### Tip
>
> You can also save your section view under the Views tab using Saved Views to quickly return to a specific cross-section without recreating the cut each time.

Work with geometry in Section View

Beyond inspection, **Section View** also allows you to interact directly with your model, making it easier to work on complex designs without needing to isolate or hide parts. When Section View is active, you can:

- Sketch on the datum plane (e.g. a construction plane) used to define the section cut
  Note: Moving or adjusting the plane will currently prevent further sketching on the updated cut.
- Measure internal features without hiding geometry
- Select and edit items more easily inside complex areas
- Modify bodies and features that are otherwise hard to access

> ### Tip
>
> To change the active section plane, select a different face or plane and turn on **Section View** again. This is a quick way to inspect your model from different orientations.

Measure

Every sketch, face, and body in Shapr3D is built with precise parameters. **Measure** lets you view the exact dimensions of elements in your model. When you turn on Measure mode, a panel appears where you can view and pin measurements for selected elements. The panel stays open while you work and can be moved anywhere on your screen.

View measurements

From the Measure Panel

Turn on Measure mode to display dimensions for selected bodies, faces, edges, or sketches.

1. From the modes in the main menu, select Measure to turn on Measure mode.
2. Make measurements of items you want to measure.
3. View the measurement of the selected items under **Pin to Always Show**.
4. Pin measurements you want to keep.

From the modeling space

For quick reference, you can select items in the modeling space to view the selection information at the bottom of the screen. The selection information includes the measurement details of the selected items.

Depending on the type of object you select, you can view the following quantitative data:

- **Length** – Length of a line. If you select multiple lines, the sum is shown.
- **Area** – Area of a face. If you select multiple faces, the sum is shown.
- **Volume** – The volume of a body. If you select multiple bodies, the sum is shown.
- **Radius** – The radius of an arc, circle, sphere, cone, or cylinder.
- **Diameter** – The diameter of an arc, circle, sphere, cone, or cylinder.
- **Angle** – The angle between any two lines or two faces. If you select an arc, its central angle is shown.
- **Perpendicular** angle – A perpendicular angle between two lines or two faces.
- **Minimum distance** – The shortest distance between two lines, arcs, circles, non-parallel faces, or bodies. It can also be the shortest distance between any combination of two of the following: circle, arc, line, face, or body.
- **Central distance** – The central distance between a center point or central axis and any of the following: line, arc, circle, parallel face, or body. Center points are calculated for circles, cones, and spheres. For bodies, the distance from the center of gravity is calculated.
- **Parallel distance** – The distance between two parallel items such as sketches, edges, and faces. Note that it is possible to calculate the distance between an edge and a parallel face. This also shows the radial difference of concentric arcs, circles, spheres, and cylinders.
- **X-axis** ΔX – The distance on the X-axis between two lines, arcs, circles, non-parallel faces, or bodies. It can also be the distance between a combination of two of the following: circle, arc, line, face, and body.
- **Y-axis** ΔY – The distance on the Y-axis between two lines, arcs, circles, non-parallel faces, or bodies. It can also be the distance between a combination of two of the following: circle, arc, line, face, and body.
- **Z-axis** ΔZ – The distance on the Z-axis between two lines, arcs, circles, non-parallel faces, or bodies. It can also be the distance between a combination of two of the following: circle, arc, line, face, and body.

Make measurements

To make measurements, select **Measure** from the available modes and then choose a measurement mode from the menu.

Object measurements

Use Object Measurements to measure the size or dimensions of a single body, face, or edge:

1. From the available modes, select **Measure**.
2. Open the Measure menu and then select **Object Measurements**.
3. Select the body, face, or edge you want to measure.
4. In the Measure pop-up panel,verify that the measurements have been added under **Pin to Always Show**.
5. **Optional**: Turn on **Include X, Y, Z Deltas** to display the distances along each axis in addition to the overall measurement.
6. **Optional:** Pin measurements that you want to keep.
7. To exit Measure mode, select **Measure** or close the Measure panel.

Point-to-point measurements

Use Point-to-Point Measurements to measure the distance between notable points:

1. From the available modes, select **Measure**.
2. Open the Measure menu and then select **Point-to-Point measurement**.
3. **Optional**: Select a face or hover over a face to find notable points.
4. Select the notable points you want to measure.
5. **Optional**: Move already placed points to new notable points.
6. **Optional**: Turn on **Include X, Y, Z Deltas** to display the distances along each axis in addition to the overall measurement.
7. **Optional:** Pin measurements that you want to keep.
8. To exit Measure mode, select **Measure** or close the Measure panel.

3-Point Angle

Use 3-Point Angle to measure the subtended angle of three notable points:

1. From the available modes, select **Measure**.
2. Open the Measure menu and then select **3-Point Angle**.
3. **Optional**: Select a face or hover over a face to find notable points.
4. Select three notable points you want to measure.
5. **Optional**: Move already placed points to new notable points.
6. **Optional**: Turn on **Include X, Y, Z Deltas** to display the distances along each axis in addition to the overall measurement.
7. **Optional:** Pin measurements that you want to keep.
8. To exit Measure mode, select **Measure** or close the Measure panel.

Highlight measurements

You can highlight measurements to see visual feedback in the modeling space and easily identify the edge, face, or point the measurement refers to. The highlight disappears when you move the cursor or tap elsewhere.

To highlight measurements:

- **Keyboard and mouse/trackpad**: Hover over a measurement in the **Measure** panel.
- **Touch and pen**: Tap a measurement in the panel.

Pin measurements

Pin measurements to keep them visible as annotations in your model. You can pin both active measurements you're currently taking and any existing measurements from the panel.

Pinned measurements stay visible while you work, making it easier to reference dimensions and communicate with others.

To pin a measurement:

1. While in Measure mode, select the **Pin**  beside a measurement in the Measure panel.
2. View your pinned measurements anytime under **Pinned** in the panel and in your model.
3. **Optional**: To delete or unpin a pinned measurement go back to the pop-up panel and select **Delete**  beside the pinned measurement.

Video walkthrough

Shapr3D Manual - Measure your models | Modes

Display Modes

You can use **Display Modes**  to control how geometry is visually presented in the modeling space. Use it to switch shading styles, run surface analysis tools, or toggle edge visibility.

Use Display Modes

To use the Display Modes menu:

1. Select **Display Modes** from the toolbar to open the menu.
2. Choose a shader, surface analysis tool, or toggle an option to update how your model appears in the workspace.
3. To check the active shader at a glance, look at the label below **Display Modes** in the menu.

Shaders

Switch between shading styles to adjust how geometry and edges appear in the workspace.

- **Wireframe** – Displays only the edges and outlines of your model, with no surface fill.
- **X-Ray** – Displays your model with semi-transparent surfaces, allowing you to see through bodies to view internal details, hidden edges, or overlapping geometry.
- **Shaded** – Displays solid shaded bodies without outlines or technical overlays. This is the default mode.
- **Visualized** – Displays your model with physically based materials in a realistic environment.

Surface Analysis

Use surface visualization tools to evaluate model continuity and curvature. These tools help with identifying surface smoothness and transitions in complex shapes.

- **Zebra** – Applies zebra striping to evaluate surface continuity (G0, G1, G2).
  - **Direction**: Choose between **Horizontal** or **Vertical** stripes.
  - **Scale**: Adjust the sensitivity of the curvature analysis display.
- **Curvature Map** – Applies a color-coded gradient to visualize curvature changes across the surface.
  - **Scale**: Adjust the sensitivity of the curvature analysis display.

Options

Toggle additional edge visibility settings to refine how your model appears in the workspace.

- **Show Edges** – Displays the edges of bodies in the modeling space.
- **Show Hidden Edges** – Displays edges that are hidden behind the model's own surfaces, making it easier to see the full geometry of a body without rotating the view.
- **Show Decals** – Shows or hides decals applied to your model surfaces in Visualization.

Collaboration

Tools and features for sharing projects, reviewing designs, and working with your team in Shapr3D
[Section link | Table of contents | Back | Next]

Team Collaboration

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Shapr3D supports team collaboration with built-in tools for sharing, reviewing, and presenting designs across devices and workflows. You can work on shared projects with teammates, track design progress, and present models in a browser or real-world scale.

Team projects

Projects stored in a shared space are automatically shared with your team. You can organize these projects into folders, assign access levels, and collaborate across devices with contributors and editors.

To learn more, visit Shapr3D Dashboard.

Project Versions

Project Versions lets you restore earlier versions of your work, making it easy to test changes or revisit past design decisions without losing progress.

To learn more, visit Project Versions.

Saved Views

Use **Saved Views** to bookmark specific angles or features of your design. You can return to these views instantly during presentations or model reviews for consistent context.

To adjust how your model is displayed, visit Views and Appearance.

Published Version

A Published Version allows you to share a 3D preview of your model with others. Individuals with access can open the link in a browser to view 3D and 2D models, and leave comments if commenting is enabled.

To learn more, visit Published Versions.

Augmented Reality (AR)

Opening a Published Version on a supported mobile device lets you see your design in Augmented Reality. This helps you evaluate scale, placement, and proportions in a real-world setting.

To learn more, visit Augmented Reality.

Vision Pro (Enterprise only)

Enterprise users can review designs using Vision Pro. Invite collaborators into an immersive workspace to explore your model in room-scale.

To learn more, explore our Vision Pro articles.

Capture tool via Visualization

Use the **Capture tool** to get snapshots of your model without the grid and other unwanted project elements. This is ideal for reviews, presentations, or marketing material. Saved images can be annotated using your device's built-in tools to highlight feedback or areas of interest.

To learn more, visit Capture images of your visualized model.

Screenshot tool

The Screenshot tool allows you to take quick snapshots of your model with adjustable visibility and appearance settings. Saved screenshots can be annotated directly on your device, making it easy to add notes and share feedback visually.

To learn more, visit Screenshot tool.

Published Versions

Published Versions let you share your 3D models and 2D Drawings through a secure web link. Viewers can open your project in any modern browser without installing Shapr3D. You can control access by sharing the link publicly, restricting it to your team, or inviting specific individuals for a more private review.

Publish current version

To publish the current version of your project:

1. From the Menu bar in the modeling space, go to the **File** menu (Windows/macOS) or the **More** menu (iPadOS), then select **Publish Current Version...** to open the **Export** dialog.
2. Under **All Formats**, select **Version**.
3. To generate your Published Version, select **Publish.**

   Note: Select ••• beside **Publish Current Version** to manage privacy and access settings before generating the link.
4. **Optional**: If Sync is disabled, you'll be prompted to enable it before continuing. Enable Sync to continue.

Manage privacy and access settings

Once the current version is published, you can manage the following settings from the **Version** tab in the **Export** dialog**:**

- **Published Version** – Copy the link to share it with selected users.
- **Link Settings** – Choose who can view the link and whether to make them a **Viewer** or **Commenter**.
  - **Anyone with the link** – Anyone can view your model.
  - **Only the people invited** – Only people invited to view your Published Version can view it.
  - **Members of this team** – Only team members logged into Shapr3D can access the model.
  - **Include Visualization Materials** – Disable this setting to speed up link creation and improve viewing performance on mobile devices.
  - **Include 2D Drawings** – Enable this setting to include 2D Drawings in the Published Version.
- **Comments** – Shows the number of comments in your Published Version. Select Open to quickly view the link and its comments.

Note: After making any changes to the Published Version settings, select **Publish Changes** to apply them. To deactivate the Published Version, select **Unpublish**.

View projects via a Published Version

View shared projects by opening the Published Version in any modern web browser. You can explore 3D models, review 2D drawings, and leave comments directly for fast, focused feedback.

3D Models

Explore shared 3D models directly in your browser with intuitive navigation and feedback tools.

- **Interact with models**: Move, rotate, and zoom in or out to explore every detail.
- **Use Augmented Reality (AR)**: Click the AR button to visualize your models in a physical environment. To learn more, visit Augmented Reality.
- **Collaborate with comments**: Team members can view and add comments directly to specific parts of the model. To learn more, go to Collaborate with comments.

2D Drawings

When 2D Drawings are included in a Published Version, a sidebar provides quick access to these 2D Drawings, making it easy to review detailed views alongside your 3D models. Here's how you can interact with and save 2D Drawings:

- **Show or hide drawings**: Click the sidebar to toggle the visibility of available 2D Drawings.
- **Interact with drawings**: Move, zoom in, or zoom out while viewing the 2D drawing.
- **Save or copy the drawing**: Right-click (secondary click) the drawing and select **Save Image As…** or **Copy Image**.
- **Download as PDF**: Click the **Download PDF** button to save a PDF copy of the drawing.

Invite users and manage access

You can control who can view or comment on your Published Version by managing access permissions. Use the **Share** button to invite users, assign roles, adjust access settings, or copy the link as needed.

Invite users to view a Published Version

You can invite specific users to access your Published Version without making the link public. Invited users will receive an email with a link to view the project. If they don't have a Shapr3D account, they'll need to sign up before accessing the Published Version.

To invite a new user:

1. Select the **Share**.
2. In the **Emails** field under **People with access**, enter the email address(es) of the person/people you want to invite.
3. From the dropdown beside, select their role:
    - **Viewer** – Can view the project but cannot comment.
    - **Commenter** – Can view and leave comments.
4. Select **Invite**.
5. To save the changes, select **Done**.

Manage Published Version access

You can control who can view or comment on your Published Version by adjusting **General access** settings. Choose between allowing anyone with the link to access the project or restricting it to team members only.

To manage access settings:

1. Select **Share**.
2. Under **General access**, choose who can access the Published Version:
    - **Anyone with the link** – Grants access to anyone who has the link.
    - **Members of this team** – Restricts access to team members only.
3. From the dropdown beside the selected option, choose a **role**:
    - **Viewer** – Can view the project but cannot comment.
    - **Commenter** – Can view and leave comments.
4. To save the changes, select **Done**.

Collaborate with comments

Published Versions let team members and invited commenters leave context-specific comments directly on your design. Commenting is enabled by default and works in desktop browsers when signed into a Shapr3D account.

Add comments

Team members and invited commenters can add comments directly when viewing the Published Version on a desktop browser while signed in to a Shapr3D account.

To add comments:

1. Open the Published Version from a desktop web browser.
2. If you haven't yet, log in to your Shapr3D account.
3. Click the **Add comment** .
4. Select any part of the model you want to comment on.
5. Enter your comment in the comment field.
6. To finish, select **Send**.

View comments

Collaborate seamlessly by viewing comments directly on the Published Version page.

- **Show or hide comments**: Click to toggle comment visibility.
- **Open specific comments**: Click on a comment to navigate to the part of the model where it was added.
- **Refresh for updates**: Reload the page to view the latest comments added by your team.

Augmented Reality

Shapr3D supports augmented reality (AR) which allows you to visualize your 3D model in the real world on your camera-equipped device by creating a virtual prototype of your project. With AR technology, you can see exactly how your project will look in its intended place.

This kind of visualization provides great feedback on different project aspects, from dimensions to aesthetics, giving you more insight into your prototype project at any point in your workflow. If you see something that could be tweaked or doesn't quite match up with what you imagined, you can make the required adjustments immediately.

Requirements

To use the AR feature, you need a device with a rear-facing camera that supports Augmented Reality. This includes most modern smartphones, all iPadOS, and a few Windows tablets.

Accessing AR

There are three main ways to use AR:

1. On iPadOS devices, you can launch an AR preview directly from the app.
2. Published the current version and open the link with your smartphone.
3. Export your project in a suitable format and use a 3rd party application.

AR previews on iPads

If you are on an iPad, AR preview can be accessed in multiple ways:

- While in Visualization, select the AR icon at the top.
- From the modeling space, access AR preview through any of the following:
    - Go to **Export** > **Usage tab** > **USDZ** > **Preview in Augmented Reality**
    - Select **Share** > **Export** > **Export Project** > **USDZ** > **Preview in Augmented Reality**

To use AR, simply aim your device's camera to a setting in which you want to visualize your model.

Use your touch to freely move and scale the model so it fits better in your environment.

You can also check the rendered model and zoom in on the details by selecting the **Object** tab at the top.

Return to the **AR** tab and select the **Shutter** button to capture the screen. The exported image will automatically save to your gallery.

Using a smartphone via a Published Version

This is the easiest way if you are designing on a laptop or a desktop computer without a rear-facing camera.

1. Generate a Published Version.
2. Open the link on your smartphone.
    - Apple Continuity, Microsoft Phone Link and Microsoft Edge Sync are features that make it very easy.
    - Alternatively, you can open the link in your desktop browser, click the **AR** button and read the QR code on the screen with your phone camera.
3. Tap the AR icon in your mobile browser.

Exporting your project and use a 3rd-party application

You can also export your projects in a suitable format and open it in 3rd-party applications that support Augmented Reality previews, such as Microsoft 3D Viewer, Apple Quick Look or Adobe Aero.

The best visual quality can be achieved through the USDZ format, currently only available on Apple platforms. 3MF is widely supported but at the moment we don’t include materials or colors set in Visualization in the export.

To learn more about exporting in different formats, visit Export.

Capture images of your visualized model

Visualization lets you present your models with realistic materials, lighting, and finishes. With over 100 material and finish types available, you can customize how your model appears to reflect the final product more accurately.

The **Capture tool** in Visualization saves high-quality images of your visualized model without gridlines or interface elements. In contrast, the **Screenshot tool** captures the modeling space and gives you the option to include visual guides and UI elements. Use the Capture tool when you need clean visuals for presentations, reviews, or client discussions.

Capture

Use **Capture** to take a direct image of your visualized model, similar to a screenshot but without gridlines or interface elements. This gives you a clean output you can save or share for presentations, reviews, or client discussions.

To capture images in Visualization:

1. Go to **Visualization**.
2. At the top, select **Capture** to get snapshots of your visualized models.
3. In the **Save** dialog, enter a filename and choose where you want to save your captured image**.**
4. Select **Save**.

Generative Render

Use **Generative Render** to enhance your captured model with AI-generated effects. To learn more, visit Generate AI renders in Visualization.

Read more

To learn more, visit Visualization.

To learn how to capture images of your model from the modeling space, visit Screenshot tool.

Generate AI renders in Visualization

Feature availability by plan

- **Shapr3D Basic**: Includes limited access for generative AI rendering. For higher limits, see available plans on our Pricing page.
- **Shapr3D Pro**: Includes expanded access to generative AI rendering. For details, visit our Pricing page.
- **Shapr3D Enterprise**: Offers full access to generative AI rendering and other advanced collaboration and data management tools. For enterprise options, contact sales.
- **Shapr3D Educational**: Generative AI rendering is not included in EDU plans. To use these features, upgrade to a Pro license or see available plans on our Pricing page.

Use **Generative Render** to enhance your captured model with AI-generated effects. You can enter a prompt describing how you'd like the image to appear and then save or share the generated result.

Note: Generative Render has a daily usage limit.

To use Generative Render:

1. Go to **Visualization**.
2. At the top, select **Generative Render**.
   Note: If Sync is disabled, you'll be prompted to enable it before continuing. Enable Sync to continue.
3. Enter a prompt describing how you want to enhance the image.
   Example: *Show this car dashboard in context, through the windshield we should see a nice pine forest inside the rocky mountains.*
4. Select **Generate**.
5. Select **Save Image** to save the generated image.

Read more

To learn more, visit Visualization.

To learn how to capture images of your model from the modeling space, visit Screenshot tool.

Screenshot tool

You can use the **Screenshot** tool to take clean snapshots of your model directly from the modeling space. Before capturing, you can customize how your model looks by adjusting visual elements like the grid, background, and body edges.

Use screenshots to:

- Capture quick visuals without UI elements
- Adjust visibility settings for clarity
- Save or copy images for presentations or feedback

How to take a screenshot

To take a screenshot:

1. Open your project in Shapr3D.
2. Go to **Export**.
3. From the available list of file formats, select **Image.**
4. Under **Image file**, select **Open Screenshot Tool…** beside **Capture Screenshot**.
5. Use the menu to customize how you want your exported image to look:
    - Grid – Remove or keep the grid in the background.
    - Transparency – Make the background transparent or filled.
    - Body Edges – Render your project with or without edges.
      Note: You can also open the Items Manager to choose which items to display.
    - **Resolution** 1x – Select the output size of your screenshot:
        - **Actual** - Captures the image at the current size of your workspace window.
        - **Double** - Doubles the pixel count of your current window for improved clarity.
        - **Full HD** - Exports at a standard 1920 x 1080 resolution.
        - **4K** - Provides high-definition output at 3840 x 2160, suitable for professional presentations.
        - **8K** - Delivers maximum detail at 7680 x 4320 for large-scale prints and high-fidelity renders.
6. To capture your model, select **Capture** .
7. In the Save dialog, enter a filename and choose where you want to save your captured image.
8. To finish, select **Save**.

Copy and reuse your last screenshot setup

The Screenshot tool lets you quickly reuse and share the same captured view without setting it up again. Here's how it works:

- The tool remembers your last used settings, including grid, transparency, and edge visibility.
- Your saved settings are applied automatically when you reopen the tool.
- You can use a keyboard shortcut to copy your last screenshot without reopening the tool.

- The copied image is saved to your clipboard, ready to paste into emails, slides, or any document tool where you need to present your design.

> **Read more**
>
> To learn how to save images of your visualized models, visit Capture images of your visualized model.

Import and export

Supported file types, exporting, and importing
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Supported file types

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Shapr3D supports a wide range of file types for easy integration with other CAD systems and collaboration. Below are the details of supported file types for importing and exporting, along with subscription-based limitations.

Importing

You can import a wide range of file formats into Shapr3D, making it easy to bring in designs from other CAD software. Whether you're importing sketches, models, or drawings, you can seamlessly work with various supported file formats.

> **Feature availability by plan**
>
> - **Shapr3D Basic**: 2D and 3D industry-standard formats, limited up to 2 projects only.
> - **Shapr3D Pro**: 2D and 3D industry-standard formats.
> - **Shapr3D Enterprise**: 2D and 3D industry-standard formats, plus advanced CAD file formats from leading CAD software like CATIA, Creo, and Solid Edge.

2D sketches

- DWG and DXF
  - 3D data: No
  - Annotation (text, dimensions, leaders, and arrows): No
  - Lines: Yes
  - Polylines: Yes
  - Bezier curves: Yes
  - Arcs: Yes
  - Circles and ellipses: Yes

- Polygons: Yes
- Names of entities: No
- Visibility: No
- Line styles (Line weight, pattern): No
- Colors: No
- Material and texture: No
- Relationship between model elements, and constraints: No
- Metadata: No

3D bodies

- X_T or X_B
- STEP
- IGES
- STL (for reference only)
- SHAPR
- SLDPRT
- SLDASM (including external references that are not stored in mesh files)

Images

- PNG
- JPG
- 2D PDF (single or first page only)
- TIFF
- BMP
- ICO
- RAW
- GIF (not animated)

Advanced CAD file formats (Enterprise only)

NX

- **Part/Assembly**: .prt

CATIA

- **Part**: .catpart
- **Assembly**: .catproduct

Creo

- **Part**: .prt
- **Assembly**: .asm

Solid Edge

- **Part**: .par
- **Assembly**: .asm, .psm

JT

- **Part/Assembly**: .jt

Exporting

Shapr3D offers flexible exporting options, allowing you to share and collaborate across different platforms. You can easily export your designs in multiple file formats, ensuring compatibility with various tools and workflows.

Feature availability by plan

- **Shapr3D Basic**: Low-resolution STL and 3MF formats only.
- **Shapr3D Pro**: 2D and 3D industry-standard formats.
- **Shapr3D Enterprise**: 2D and 3D industry-standard formats, plus advanced CAD file formats from leading CAD software.

Sharing options

- Projects
- Published Versions

Tip

Save time by sharing instead of exporting:

- Use **Project** to collaborate on the same design file without emailing .shapr files back and forth.
- Published the current version to send a view-only version of your model or drawing in the browser — no need to export PDFs, DXF, or AR files separately.

2D

- DWG (sketches and 2D Drawings)
- DXF (sketches and 2D Drawings)
- PDF
- JPEG
- PNG
- SVG
  Note: When exporting sketches, unsupported drafting elements such as layers, hatches, and annotations are filtered out.

3D

- SHAPR (the native file format of Shapr3D)
  Note: If you have Sync disabled, you can use our native SHAPR format to transfer models between different Shapr3D platforms.
- X_T or X_B
- STEP
- 3MF

- IGES
- OBJ (with colors)
- STL
- GLB
- USDZ

Advanced CAD file formats (Enterprise only)

JT

- **Part/Assembly**: .jt

Import

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You can import a wide range of file formats into Shapr3D, allowing you to easily bring in designs from other CAD software. From sketches to 3D models and technical drawings, this article guides you through the process of importing supported formats.

Before you begin

- Ensure that you're importing supported file types.
- Verify that the source file is not damaged or corrupted.
- Confirm that your device has enough memory to process the import.
- The source model should fit within Shapr3D's design space, which is 1 km³.
- Avoid importing files that include sketches with polylines.
  Note: Some lines and curves from other software may not display accurately in Shapr3D.

Import Preferences

You can also enable advanced import preferences in Settings or Preferences to customize how Shapr3D handles incoming geometry. These options allow for faster processing and better model quality when importing STEP, IGES, CATIA, SolidWorks, and Solid Edge files.

The following advanced settings are available during the import process:

- **Simplify Geometry** – Remove redundant topology, simplify smaller details. This might change the model's shape.
- **Advanced Healing (Parasolid Bodyshop Repair)** – Precisely recalculate all edges based on face intersections and replace tolerant edges with accurate ones.
- **Healing (HOOPS)** – Controls adjustment of topological tolerances, removing geometric discontinuities, eliminating slicer faces, etc.
- **Accurate Edge Computation** – Analyze bodies and attempt to rectify any geometry problems found.
- **Sewing** – Joins touching surfaces during import to form valid bodies. Improves editability and downstream modeling results.
- **Import Planar Curves as Sketches** – Convert planar curves (circles, lines, splines) into editable sketches in History.

Import images

Images help anchor your design process by serving as visual guides while you model. Import photos, sketches, or scanned drawings when you need a reference to trace, match proportions, or align design features.

To import an image:

1. From the **Insert menu**, select **Image**.
   Note: You can also access **Import** from the **File menu** (Windows/macoS), **More menu** (iPadOS) or by selecting **Import** in the Dashboard.
2. Choose a photo, file, or if using an iPad, capture a new one with your device's camera.
3. Use the gizmo to scale, move, or rotate the image.
4. Adjust **Opacity** to help with tracing or overlaying details.
5. To finish, select **Done**.

Tip

- **For macOS**: You can also drag and drop a file or image into the Shapr3D icon in the dock on your Mac desktop.
- **For iPadOS**: If you want to import images from the **Photos** app, you may need to allow Shapr3D access to the photos on your device. To grant access, go to **System Preferences > Security & Privacy**.

Import files

External CAD files and existing Shapr3D projects can be brought into your workspace when you need to continue work started elsewhere or combine multiple designs. Importing allows you to integrate geometry from other CAD tools, reuse parts from previous Shapr3D projects, or incorporate supplier components directly into your current model.

To import a file:

1. From the **Insert** menu, select **File**.
   Note: You can also access **Import** from the **File menu** (Windows/macoS), **More menu** (iPadOS) or by selecting **Import** in the Dashboard.
2. Choose a file to import.
3. **Optional:** Use the Move/Rotate or Translate tool to reposition your newly imported file.

Import Shapr3D projects from the Dashboard

You can bring entire Shapr3D projects into your current design directly from the Dashboard. Importing existing projects into your current one allows you to reuse models you've already created, combine multiple projects into one workspace, or build larger designs by extending earlier work. Once imported, the project appears in the Items Manager, and its features are added to your project's History so you can continue editing everything in a single, unified model.

To import a Shapr3D project from the Dashboard:

1. From the **Insert menu**, select **Project**.
2. Choose a Shapr3D project from your Dashboard.
   Note: To select and import multiple projects at once, tap Select (iPadOS), or hold down CTRL (Windows) or Shift (macOS).
3. Select **Import**.

History settings

When you import files or projects, Shapr3D adds the action to your History so you can review or adjust how the imported content is added to your design.

How imported items appear in History depends on the file type:

- **Shapr3D projects (.shapr) with unmerged history steps**
  These projects keep their original history. Their feature steps appear in your History sidebar, allowing you to continue editing them as if they were created in your current project.
- **All other imported formats (STEP, IGES, STL, DWG, DXF, images, etc.)**
  These are added as a single Import step in History. The geometry is still fully editable, but the import action is treated as one item instead of multiple feature steps.

To manage an import entry in History:

1. Open **History**.
2. Look for the import step you want to edit.

Click or tap **Settings (…)** to modify the history settings for the import step.

> ### Tip
>
> Files you import are automatically added to your Items Manager. If you import a 3D model in STEP format, the hierarchical structure of the source model is retained in Shapr3D. Bodies that were grouped together in the same unit (assemblies, parts, components, etc.) are organized into nested folders.

Export

You can export your designs to other software, collaborate seamlessly with your team, or bring your models into Apple's AR ecosystem for immersive visualization. Whether you need to gather feedback or prepare for production, Shapr3D gives you the flexibility to move your designs smoothly across platforms.

Before you begin

- Ensure that you're exporting supported file types.

Export your projects

To export a file, follow these steps:

1. In the modeling space, access the file menu (macOS/Windows) or more menu (iPadOS), then select **Export to** > **Export...**
2. Choose your preferred format from the available format types.
3. **Optional**: To export in batches, click/tap **Select** and select the formats you need.
4. If applicable, customize your project details.
5. Select **Export**.
6. **Optional**: Save your favorite formats before exporting to make future exports quicker and more convenient.
7. In the **Save As** dialog, enter a file name and choose a location where you want to save your exported file.
8. Select **Save**.

Exporting with isolated parts

You can export isolated parts in any file type except the default **.shapr** format. Isolated exports to **.shapr** are unavailable to ensure the complete design is preserved in this format.

To export isolated parts:

1. **Isolate** the part you want to export.
2. Follow the steps to export your projects.

File format types

When exporting, you can select from the list of available formats or choose from specific categories:

> **Tip**
>
> Save time by sharing instead of exporting:
>
> - Use **Project** to collaborate on the same design file without emailing .shapr files back and forth.
> - Publish the current version to send a view-only version of your model or drawing in the browser — no need to export PDFs, DXF, or AR files separately.

- **All formats** – View all recommended and supported formats.
- **Share options**:
  - **Project** – Manage access and share a project link with details that can be viewed in a browser or opened in Shapr3D. You can also download the .shapr file. To learn more, visit Share project details to your team.
  - **Published Version** – Publish and share a link to view your 3D models and 2D drawings in a browser without installing Shapr3D. Viewers can preview, review, and comment depending on permissions. To learn more, visit Published Versions.
- **3D model** – Export formats compatible with 3D models, including **Shapr3D native format**, **Parasolid**, **STEP**, **IGES**, **3MF**, **STL**, and **OBJ**.
- **3D print** – Export formats for 3D printing, such as **STL** and **3MF**.
- **Sketch** – Export sketch files, including **DWG**, **DXF**, and **SVG**.
- **Drawing** – Export drawing files, including **DWG**, **DXF**, and **PDF**.
- **AR** – Export formats for augmented reality, such as **USDZ** and **GLB**.
- **Image** – Export images in **PNG** and **JPEG** formats.
- **Slicer (by Prusa)** – Send your design directly to PrusaSlicer or EasyPrint to prepare prints for your Prusa 3D printer. To learn more, visit Export for 3D Printing to PrusaSlicer in Shapr3D.
- **Slicer (Bambu)** – Send your design directly to Bambu Studio on desktop to prepare prints for Bambu Lab printers. To learn more, visit Export for 3D Printing to Bambu Lab in Shapr3D.
- **Slicer (Orca)** – Send your design directly to OrcaSlicer on desktop to prepare prints for a wide range of FDM printers. To learn more, visit Export for 3D Printing to OrcaSlicer in Shapr3D.

Project details

You can customize your projects when exporting with the following details:

- **File name** – Enter a name for your project.
- **File Format/Output Format** – You have several options: AP203, AP214, AP242, text, ASCII, and binary. The latter is the default format since it's more common and compact.
- **Include Dimensions** – Any locked dimensions in your project will be preserved.
- **Include Hidden Items** – Turn on the switch if you want hidden items to be saved in the export.
- **Include Mesh Bodies** – Turn on the switch if you want mesh bodies to be saved in the export.
- **Compress Geometries** – Turn on to compress geometries in your projects.
- **Compress Textures** – Turn on to compress textures in your projects.
- **Include Hidden Sketches** – Turn on the switch if you want hidden sketches to be saved in the export.
- **Include Vertex Colors** – Turn on the switch if you want to preserve vertex colors in the export.

- **Resolution** – Set the quality of the export. If you want to set a custom resolution, you can set the deviation and angle tolerances.
- **Save Each First-Level Item Separately** – Turn on the switch if you want to export your project in separate, organized files. These files are then compressed into a ZIP file.
- **Save Each Sketch Plane Separately** – Turn on the switch if you want to export your project in separate, organized files. These files are then compressed into a ZIP file.
- **Units** – Set the unit of your export to millimeter, centimeter, meter, inch, or foot.

Save your favorite formats

You can save your favorite formats during export to make exporting quicker and more efficient.

To favorite a format, hover over a file format and then select the star symbol.

Video walkthrough

Shapr3D Manual - Export your projects

Export to a 3D printer slicer in Shapr3D

Shapr3D integrates directly with a range of industry-standard slicers, letting you send your design from the Export menu to your slicer of choice without any manual file handling. No need to export a file, locate it, and import it separately — the integration handles the transfer automatically and opens your model in the slicer, ready to prepare for printing.

Video walkthrough

https://www.youtube.com/playlist?list=PLYYzpLiuoIXc

General workflow

Exporting to a slicer follows the same general steps across all supported integrations.

1. Open the Export menu and choose the 3D Print category to display the available slicer integrations.
2. Select your slicer.
3. Your model is transferred and opened automatically in the slicer.
4. Set up your print settings, supports, and slicing parameters directly in the slicer.

Before you export

Review the following before sending your model to a slicer.

- Your model is complete and ready to print. Any changes made after exporting will require a new export.
- All bodies are in the correct position and orientation. What you see in Shapr3D is what gets sent to the slicer, including scale and placement.

Supported slicers

Shapr3D currently supports direct integration with a selection of widely used slicers. Each integration follows the same general export flow, but the options and settings available will vary depending on the slicer. For step-by-step instructions, visit the article for your slicer:

- PrusaSlicer and Prusa EasyPrint
- Bambu Studio
- OrcaSlicer

Export for 3D printing to PrusaSlicer in Shapr3D

Video walkthrough

https://www.youtube.com/playlist?list=PLByj0sWIAgGM

Shapr3D integrates directly with Prusa's slicing tools, so designs can be sent straight from the export menu to either Prusa EasyPrint or PrusaSlicer. The Slicer (by Prusa) export option works with a range of 3D printers, making it easy to go from model to print without manual file transfers.

Before you begin

To export to a slicer, make sure you have one of the following:

- A free Printables account to use Prusa EasyPrint. EasyPrint works with a range of 3D printers, so designs can be sent directly to your own printer after slicing.
- PrusaSlicer installed on your desktop to use PrusaSlicer

Export to a 3D printer slicer

1. Go to **Export** and select **Slicer (by Prusa)** from the format list.
2. Adjust the export settings as needed. See Slicer export settings below.
3. Choose how to open your file:
    - Select **Open in EasyPrint** to send the file to Prusa EasyPrint in the browser for instant slicing.
    - Select **Open in PrusaSlicer** to send the file to the PrusaSlicer desktop app for full slicing control.

Slicer export settings

Before exporting, you can adjust the following settings:

- **Resolution** – Controls the mesh quality of the exported file. Choose from **Low**, **High**, or **Custom**.
- **Include Mesh Bodies** – When on, mesh bodies in your design are included in the export. On by default.
- **Include Hidden Items** – When on, hidden items in your design are included in the export. Off by default.
- **File Name** – Set a custom name for the exported file. Defaults to the project name.

EasyPrint vs PrusaSlicer

Both options send your file directly to Prusa's slicing tools, but serve different workflows.

Prusa EasyPrint

Prusa EasyPrint is a free browser-based slicer available on Printables. No installation needed, but requires a free Printables account.

Once your Printables account is set up:

1. Select your printer.
   Note: For first-time setup, follow the Prusa guide.
2. Choose a material.
3. Adjust print settings.
4. Send directly to your printer.

PrusaSlicer

PrusaSlicer is a full-featured desktop slicer with advanced print settings and full slicing control. Best for complex prints, custom printer configurations, or when you need fine-tuned control over slicing parameters.

To use PrusaSlicer:

1. Open your file in PrusaSlicer.
2. Configure your printer, material, and slicing settings.
3. Slice the model and export or send to your printer.

Export for 3D printing to Bambu Lab in Shapr3D

Shapr3D integrates directly with Bambu Studio, so designs can be sent straight from the Export menu to your slicer without manual file transfers. The **Slicer (Bambu)** export option works with the range of Bambu Lab printers, making it easy to go from model to print in just a few steps.

Video walkthrough

https://www.youtube.com/playlist?list=PLE7WJeOF62ck

Before you begin

To export to Bambu Studio, make sure you have the following:

- Bambu Studio installed on your desktop.

Export to Bambu Studio

1. Go to **Export** and select **Slicer (Bambu)** from the format list.
2. Adjust the export settings as needed. See Slicer export settings below.
3. Select **Open in Bambu Studio** to send the file to the Bambu Studio desktop app.

Slicer export settings

Before exporting, you can adjust the following settings:

- **Resolution** – Controls the mesh quality of the exported file. Choose from Low, High, or Custom.
- **Include Mesh Bodies** – When on, mesh bodies in your design are included in the export. On by default.
- **Include Hidden Items** – When turned on, hidden items in your design are included in the export. Off by default.
- **File Name** – Set a custom name for the exported file. Defaults to the project name.

Using Bambu Studio

Bambu Studio is a full-featured desktop slicer with advanced print settings and full slicing control. Best for complex prints, custom printer configurations, or when you need fine-tuned control over slicing parameters.

To use Bambu Studio:

1. Open your file in Bambu Studio.
2. Configure your printer, material, and slicing settings.
3. Slice the model and export or send to your printer.

Export for 3D printing to OrcaSlicer in Shapr3D

Shapr3D integrates directly with OrcaSlicer, so designs can be sent straight from the Export menu to your slicer without manual file transfers. The **Slicer (Orca)** export option is compatible with a wide range of FDM printers, making it easy to go from model to print in just a few steps.

Video walkthrough

https://www.youtube.com/playlist?list=PLFqdo2Dtyf_M

Before you begin

To export to OrcaSlicer, make sure you have the following:

- OrcaSlicer installed on your desktop.

Export to OrcaSlicer

1. Go to **Export** and select **Slicer (Orca)** from the format list.
2. Adjust the export settings as needed. See Slicer export settings below.
3. Select **Open in OrcaSlicer** to send the file to the OrcaSlicer desktop app.

Slicer export settings

Before exporting, you can adjust the following settings:

- **Resolution** – Controls the mesh quality of the exported file. Choose from Low, High, or Custom.
- **Include Mesh Bodies** – When on, mesh bodies in your design are included in the export. On by default.
- **Include Hidden Items** – When turned on, hidden items in your design are included in the export. Off by default.
- **File Name** – Set a custom name for the exported file. Defaults to the project name.

Using OrcaSlicer

OrcaSlicer is a full-featured open-source desktop slicer compatible with a wide range of FDM printers. Best for complex prints, custom printer configurations, or when you need fine-tuned control over slicing parameters.

To use OrcaSlicer:

1. Open your file in OrcaSlicer.
2. Configure your printer, material, and slicing settings.
3. Slice the model and export or send to your printer.

Assemblies

An assembly lets the user define specific relations between bodies in the modeling space based on geometries. For example, if there are two parts that have holes, it's possible to set one of the holes on each part to be co-axial. With an assembly, a dynamic relationship is set between the parts.

Shapr3D doesn't support assemblies yet, but as a workaround, you can use the **Align** tool to position parts together. However, when you move the parts away from their point of connection, they become independent from each other, unlike in assemblies where the relationship between the parts is dynamic where all the connected parts move together.

Shapr3D supports loading assemblies created in Solidworks, but keep in mind that the dynamic property of an assembly will be lost upon importing it.

Importing a Solidworks assembly file to Shapr3D

The methods below show you how to save a Solidworks assembly file and import it to Shapr3D.

Directly importing the main SLDASM file to Shapr3D

Refer to Solidworks' File Menu (SOLIDWORKS Visualize Connected) to learn how to export the main SLDASM file, and then follow Import.

The parts that are used in an assembly are regular CAD models called *part* in Solidworks with a file extension of SLDPRT. These parts can be located anywhere in the storage system. The parts have a link that helps locate them from anywhere in the folder structure of the given storage system. The link is a standalone file, which is the assembly file with the extension of SLDASM.

To make sure the assembly can be transferred properly, the link (SLDASM) and the parts (SLDPRT) should be transferred altogether.

Save the assembly file as a Parasolid format (x_t)

Refer to Solidworks' **Parasolid Export Options** to learn how to save a Solidworks assembly file as a Parasolid format, and then follow Import.

Save the assembly file as a STEP format

Refer to Solidworks' **STEP Export Options** to learn how to save a Solidworks assembly file as a STEP format, and then follow Import.

Save the assembly file as a virtual component

Saving the assembly file as a virtual component is the most secure method because it stores all subassemblies and parts internally in one file. However, this method might require you to change the way you organize your assemblies in Solidworks.

Refer to Solidworks' **Saving Virtual Components to External Files** to learn how to save a Solidworks assembly file as a virtual component, and then follow Import.

Mesh files

Mesh models are made up of polygons that are a collection of vertices, edges, and planar faces. Unlike CAD models created in Shapr3D, mesh models don't have curved edges or faces. However, a higher number of polygons in mesh models can result in a smoother and more curved look.

You can create a mesh model of a CAD design by converting the CAD model within a given tolerance. This is the maximum allowed distance between the CAD model and the mesh file. Smaller tolerances need more polygons and result in a more accurate model.

Note: In Shapr3D Basic, the tolerance of mesh models is loose with low-quality export which can result in reduced modeling details. While in Shapr3D Pro, tolerances are exported in high-quality and you can also customize tolerance settings.

Mesh models are used widely in the manufacturing industry but their editability has a lot of differences compared to a CAD model.

Working with mesh files in Shapr3D

You can use mesh files as a reference for your project with solid bodies, or modify it with boolean tools, transformations, and Visualization.

To use mesh data, save it in a mesh file such as STL, and import it into Shapr3D.

In this example, we imported a knob into Shapr3D and used the Extrude tool on guide sketches to cut away from the mesh body. We then used the Subtract tool to remove the unwanted extruded bodies, resulting in the desired final shape.

> **Tip**
>
> You can only use boolean operations with closed (solid) mesh bodies, not open (sheet) mesh. Regardless of the type of boolean operation you use with a mesh body, the body will always stay a mesh body.

STL units

The STL file format is a widely used mesh format in 3D printing and rapid prototyping. An STL file only contains the geometry as a polygonal mesh model, so it takes up little storage space. However, the geometry is transferred in a unitless format, making it necessary to define the expected unit when exporting and importing STL files. This characteristic of STL files can cause limitations when preparing models for 3D printing or transferring STL files to other software.

For example, if a model is created in inches and the slicer uses millimeters by default, a possible workaround is to scale the model in the slicer or standardize the unit of measurement in the entire workflow.

In Shapr3D, you can change the unit of measurement of your model to the unit of your slicer during the export process.

Read more

To learn more about exporting files, visit Export.

CNC cutting

CNC cutting is a widely used manufacturing method where a certain shape is cut from a sheet of material such as wood, metal, or plastic. CNC cutting begins after a set of steps, called *toolpath,* is generated from a group of planar curves.

In most cases, a laser, waterjet, or plasma cutter does the cutting depending on the applied material and the desired finishes. The shape the machine will follow can be created in many different ways, each with its own set of advantages and disadvantages.

Export your model for CNC cutting

In Shapr3D, you can create the contour for generating the toolpath for CNC cutting by creating a layout of the 3D bodies of your model. Follow this guide to learn how.

Step 1: Create a copy of your 3D bodies

Create a copy of the 3D bodies you want to export for CNC cutting. This will be the body you can manipulate to create your sketches that can be exported for the toolpath.

To copy a body:

1. Select the body and then turn on the **Copy** badge.
2. Using the gizmo, drag the arrow to create the copied body.
3. To finish, select an empty area of the grid.

Step 2: Move the copied bodies to a new Items folder

For easy visibility, move your copied bodies to a new **Items** folder.

To move items to a new folder:

1. Select your copied body.
2. Create a new folder with your desired name in the Items Manager.
3. Drag the selected items to the new folder.

Step 3: Hide your original bodies

Hide your original bodies in the **Item Manager** to make it easier to focus on your copied bodies.

To hide items:

1. Open Items Manager.
2. Click/tap on the visibility icon beside each body you need to hide.

Step 4: Create a closed reference sketch planar to the XY plane

Draw a closed sketch that is planar to the XY plane (Top view). The closed sketch serves as your reference for repositioning the bodies.

To draw a reference sketch planar to the XY plane:

1. Activate the XY planar view by pressing the **Space bar** key or clicking **Top** on the Orientation Cube.
2. Use the Rectangle or Circle tool to draw a reference closed sketch on the offset plane.
3. To finish, select **Exit Sketching**.

> **Important**
>
> Some 2D tools can only recognize planes that are parallel to the X-Y plane. If the exported curves are aligned to a different plane direction, the contour may be distorted or completely missing.

Step 5: Align your bodies to your reference sketch

Use the **Align** tool or **Move/Rotate** tool to lay or align the parts of your bodies to your reference sketch.

To align parts of your bodies using the **Align** tool:

1. Select a part you want to align.
2. From the adaptive menu, select **Align**.
3. Select your reference sketch to automatically align the part.
4. Use the arrows to organize the part to your desired layout.
5. To finish, select **Done**.
6. Repeat the steps for the remaining parts.

To align parts of your bodies using the **Move/Rotate** tool:

1. Select a part you want to align.
2. From the adaptive menu, select **Move/Rotate**.
3. Using the gizmo, move the part to align it to your reference sketch.
4. Organize the part to your desired layout.
5. To finish, select an empty area of the grid.
6. Repeat steps 1 to 3 for the remaining parts.

Step 6: Hide your reference sketch

Once your parts are organized according to the reference sketch, you can hide the reference sketch.

To learn how to hide a sketch or item, visit Items Manager.

Step 7: Add an offset plane for your repositioned parts

Use the **Construction Plane** tool to add an offset plane from your repositioned parts. The offset plane will serve as a reference for the next step.

To add an offset plane:

1. Select the top face of any part from your repositioned parts.
   Note: For easy reference, you can choose a part located at the center of your layout.
2. From the adaptive menu, select **Add Plane**.
3. Use the arrows to drag your offset plane upwards.
   Note: In the next step, you will project your parts to the offset plane so ensure that there's enough distance between your parts and the offset plane.
4. To finish, select an empty area of the grid.

Step 8: Project the bodies to the offset plane

Use the **Project** tool to project the top faces of your parts as sketches to the offset plane. These projected sketches will be the reference for your toolpaths for cutting.

To project the top faces of bodies to a plane:

1. Select the top faces of all your parts.
2. Select the offset plane.
3. From the adaptive menu, select **Project**.
   Note: The selected face to be projected are highlighted in blue while the offset plane where the items are to be projected is highlighted in purple.
4. To finish, select **Done**.

> **Important**
>
> Make sure to only select the top faces for the projection. Projecting an entire body results in duplicate curves which may not be handled properly.

Step 9: Show your original bodies and hide unnecessary items

Now that you have your projected sketches, you can show your original bodies again and hide items you no longer need.

From the Items Manager:

- Hide the offset plane, your repositioned/copied bodies, and other unnecessary items.
- Show your original bodies.
- You can also rename your projected sketches for easy reference.

Step 10: Isolate your projected sketches

Isolate your projected sketches to prepare them for export.

To isolate your sketches:

1. From the **Items Manager**, select your projected sketches.
2. From the modes menu, select **Isolate**.
3. To finish, select an empty area of the grid.

You're now ready to export your sketches for your toolpaths.

Step 11: Export your projected sketches

Export your projected sketches in a DXF, DWG, or SVG format.

To export your sketches:

1. From the menu, select **Export**.
2. Go to the **Sketch** tab.
3. Select your preferred sketch format.

4. Save the exported file to your desired location.
   *To learn more about exporting, visit **Export**.*

You're now ready to generate your toolpaths.

> **Important**
>
> You can create a drawing in 2D Drawings to scale your toolpaths to a specific sheet size. However, we recommend that you use the Sketch Export tool if you don't need blueprints.

Step 12: Generate your toolpath from your exported file

Load your exported sketch file to your CAM software or CNC machine to generate the toolpath.

Shapr3D Beta

Installing, updating, submitting feedback, and opting out of beta
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Shapr3D Beta features

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Shapr3D Beta is a test version of Shapr3D that you can install via direct download or Apple TestFlight invite link. With Shapr3D Beta, you can test new and upcoming features and provide us feedback about your experience. Feedback we receive allows us to make improvements to these upcoming features, ensuring a great user experience once the final version is released to Shapr3D.

Shapr3D currently has the following feature(s) available for beta testing:

- History-Based Parametric Modeling – Download here

To learn how to use this beta feature, visit the following articles:

- Install Shapr3D Beta
- Update Shapr3D Beta
- Submit beta feedback
- Opt out of Shapr3D Beta

Install Shapr3D Beta

You can install Shapr3D Beta directly from the Shapr3D website download page. Occasionally, Shapr3D sends out invite links via email that allows you to directly download Shapr3D Beta.

> **Tip**
>
> To learn about the current features available for beta testing, visit Shapr3D Beta features.

Follow this guide to learn how to install Shapr3D Beta.

Windows

Step 1: Download Shapr3D Beta

To download Shapr3D Beta:

1. Go to the Shapr3D website download page.
2. Under **Windows**, click **Download for Windows** to download the installer.
3. Once the download is complete, open the installer to install Shapr3D Beta.

Step 2: Start testing features in Shapr3D Beta

Open **Shapr3D Beta** from **TestFlight** to start using it just how you would normally do in Shapr3D.

Here are some tips when using Shapr3D Beta:

- If you're using an invite-only beta version, make sure to log in with your email address that got the invite to use Shapr3D Beta.
- The three-device limit includes both Shapr3D and Shapr3D Beta so if your account is logged in to three devices, the one with the oldest login date will be logged out once you log in to Shapr3D Beta.
- To start modeling with your current projects, export them from Shapr3D and import them into Shapr3D Beta.
- Once you edit a project in Shapr3D Beta, it becomes incompatible when you import it back to Shapr3D. To prevent this issue, keep a copy or backup of your projects with the current version of Shapr3D.
- Shapr3D Sync works separately for Shapr3D and Shapr3D Beta. For example, if you sync your projects in Shapr3D, you can only access those synced projects when you log in to Shapr3D on another device.

iPadOS/macOS

Step 1: Download TestFlight

Before you can install **Shapr3D Beta** , you must have **TestFlight** installed on your device. TestFlight is the service that Apple uses to install beta versions of software and applications.

To download TestFlight:

1. Go to Apple TestFlight to learn about the required platforms and available languages.
2. Open **App Store**. Make sure you're logged in to your Apple ID.
3. Search for **TestFlight.**
4. Tap **Get** to download.

Step 2: Download Shapr3D Beta

To download Shapr3D Beta :

1. Go to the Shapr3D website download page.
2. Under **iPadOS & macOS**, select **Download for TestFlight** to get directed to the TestFlight guide page.
3. Under **Step 2** (Join the Beta), select **Start Testing**.
4. If your browser prompts you to open TestFlight, select **Open TestFlight**.
5. From the TestFlight app, select **Shapr3D Beta**.
6. Select **Install** to download**.**

Step 3: (Optional) Turn off automatic updates

Turning off automatic updates in Shapr3D Beta is encouraged to ensure that you can keep a copy or backup of your projects.

To turn off automatic updates:

1. Open **TestFlight**.
2. Select **Shapr3D Beta** .
3. Under **App information or App Settings**, turn off **Automatic Updates**.

Step 4: Start testing features in Shapr3D Beta

Open **Shapr3D Beta** from **TestFlight** to start using it just how you would normally do in Shapr3D .

Here are some tips when using Shapr3D Beta :

- If you're using an invite-only beta version, make sure to log in with your email address that got the invite to use Shapr3D Beta .

- The three-device limit per account includes both Shapr3D and Shapr3D Beta so if your account is logged in to three devices, the one with the oldest login date will be logged out once you log in to Shapr3D Beta .
- To start modeling with your current projects, export them from Shapr3D and import them into Shapr3D Beta .
- Once you edit a project in Shapr3D Beta , it becomes incompatible when you import it back to Shapr3D . To prevent this issue, keep a copy or backup of your projects with the current version of Shapr3D .
- Shapr3D Sync works separately for Shapr3D and Shapr3D Beta . For example, if you sync your projects in Shapr3D , you can only access those synced projects when you log in to Shapr3D on another device.

Update Shapr3D Beta

Ensure that you have all the latest beta features available by keeping your Shapr3D Beta software version up-to-date. Follow the guide below to learn how.

> **Tip**
>
> To learn about the current features available for beta testing, visit Shapr3D Beta features.

Windows

Shapr3D Beta updates automatically on your device and will run continuously until the final version is released to Shapr3D.

iPadOS/macOS

Before you update Shapr3D Beta, keep a copy or backup of your projects to ensure that you don't lose your previous projects if the new update doesn't support those older projects.

If you have automatic updates turned on in TestFlight, Shapr3D Beta automatically updates to the new version and you don't need to do anything.

If you have automatic updates turned off in TestFlight, follow these steps to update Shapr3D Beta:

1. Open **TestFlight**.
2. Select **Shapr3D Beta**.
3. Select **Update**.
4. To start using Shapr3D Beta right away, select **Open**.

Submit beta feedback

Submitting feedback about your Shapr3D Beta experience is highly encouraged so we can take note of any improvements and fixes to ensure a great user experience once the final version is released to **Shapr3D** .

You can submit beta feedback in the following ways:

Submit feedback via email

Send an email directly to beta@shapr3d.com with the following information and/or attachments:

- **Subject line** – Enter a relevant subject line relevant to your feedback.
- **Description of the issue or bug** – Describe the issue you experienced such as what tools or workflow you were doing when the issue occured.
- **Relevant file attachments**
  - **Visuals** – Attach visual assets such as screenshots (Windows, iPadOS, macOS) or screen recordings (Windows, iPadOS, macOS) that visualizes the issue you experienced.
  - **Project file (preferrably in .shapr format)** – Export your project with the issue and attach it to your email.
- **System or Device information**
  - **Windows** – Provide the model version, CPU (processor), and GPU information of the device to help us identify device-specific issues.
  - **iPadOS/macOS** – Provide the device type, year, and memory information of your iPadOS or macOS device to help us identify device-specific issues.
    Note: You can take a screenshot of the device information pages and include them with the other attachments.

Submit feedback via Shapr3D Community

Go to Shapr3D Community and create a new post or comment to share your feedback.

Note: A Shapr3D Community account is different from a Shapr3D account. If you don't have a Shapr3D Community account, select **Sign Up** at the top-right to create an account.

To create a new post in Shapr3D Community:

1. Go to Shapr3D Community, then log in with your account.
2. On the top-right corner, select **New Topic**.
3. From the title field, enter a title relevant to your feedback.
4. From the category drop-down, choose the topic about the beta feature you're testing. Other common topics you can choose from are:
   - Need help? We are here. – Posts about questions you have about Shapr3D or issues you encounter that you need help with.
   - Feature requests – Posts about your suggestions, feedback, or comments about possible features that you think Shapr3D should implement in the future.
5. Enter the following information and/or attachments to your post:
   - **Subject line** – Enter a relevant subject line relevant to your feedback.
   - **Description of the issue or bug** – Describe the issue you experienced such as what tools or workflow you were doing when the issue occured.

- **Relevant file attachments**
    - **Visuals** – Attach visual assets such as screenshots (Windows, iPadOS, macOS) or screen recordings (Windows, iPadOS, macOS) that visualizes the issue you experienced.
    - **Project file (preferrably in .shapr format)** – Export your project with the issue and attach it to your email.
- **System or Device information**
    - **Windows** – Provide the model version, CPU (processor), and GPU information of the device to help us identify device-specific issues.
    - **iPadOS/macOS** – Provide the device type, year, and memory information of your iPadOS or macOS device to help us identify device-specific issues.
      Note: You can take a screenshot of the device information pages and include them with the other attachments.
- **Current Window mode** – Let us know if you're using Shapr3D Beta in a **full screen** or **windowed mode**.

6. To add your attachments, simply drag and drop them to your draft.
7. Once you're ready to post, select **Create Topic**.

Opt out of Shapr3D Beta

You can opt out of Shapr3D Beta at any time. If you decide to opt out, make sure to keep a copy or backup of your projects in case you want to access them in Shapr3D once the beta version is officially released.

Windows

To opt out of Shapr3D Beta, simply uninstall it just as you would in any normal app or program.

iPadOS/macOS

To opt out of Shapr3D Beta:

1. Open **TestFlight**.
2. Select **Shapr3D**.
3. Select **Stop Testing**.
4. Follow the on-screen instructions to be removed from the beta pool.

Once you opt-out of testing, **TestFlight** and **Shapr3D Beta** remain installed on your device. If you no longer need them, you can uninstall them (iPadOS, macOS) just how you would in any normal apps or programs.

Get help

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Submit a support request

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Depending on your subscription type, you can submit a request or open a support ticket for the following issues:

- **Shapr3D Pro**: Priority technical support for software, account, subscription, or troubleshooting issues.
- **Shapr3D Basic and Shapr3D Educational:** Shapr3D issues such as crashes and data loss.

> **Tip**
>
> For questions about modeling or creating projects, check out our Help Center or visit our Shapr3D Community and social channels.

Create a support ticket

Requests you submit automatically open a support ticket that is received by our support team. An available support representative will get back to you via email once they receive your request.

> **Tip**
>
> **In case of a project crashing, or completely or partially not opening, the support team can try to revert your project if you have Sync enabled**. As long as Sync is enabled, your work is not completely lost. If you submit a request and provide Shapr3D consent to recover your work, will do everything to restore your project(s)our support team can easily help restore your project(s).

To submit a request,

1. Go to Shapr3D Support.
2. Click **Log in** to view all issue options.
3. From the dropdown menu, choose an issue type.
   Note: If you're unable to log in or have password reset-related issues, select the available issue type about account access.
4. Fill out the form with all the required information. The basic information you need to provide are the following: **Subject**, **Description**, and **Attachment**. See **Information to provide in support tickets** below to learn what details you should provide depending on your issue.
5. Once you've filled out all the information needed, click **Submit**.
6. Wait for a support representative to get back to you via the email you provided.

Information to provide in support tickets

Subject

In the subject field, enter a subject that best summarizes your issue.

Description

In the description box, provide a brief description of your issue or concern. Follow the provided guidelines indicated below the description box on what not to include in your request:

- **Your consent that we can take a look at your data.**
- Your Shapr3D app version found under Settings or Preferences.
- The detailed hardware and software specs of the machine you are using.
- A detailed description of what leads to the problem.
- A screen recording of the issue. For more information, see Attachments.
- The Shapr3D project in .shapr format for which you're experiencing the issue. For more information, see Attachments.
- The name of the project where you're experiencing the issue.

In case of a project crashing, or completely or partially not opening, the support team can try to revert your project if you have Sync enabled. As long as Sync is enabled, your work is not completely lost. If you submit a request and provide Shapr3D consent to recover your work, our support team will do everything to help restore your project(s).

Attachments

Attachments are optional, but we recommend that you submit a screenshot or screen recording for specific issues to help our support team collect information that can help resolve your issue faster.
Note: Check out Create screen recordings for your request to learn how to attach regular and larger files.

> **Important**
>
> If you choose to provide attachments of your work in your support ticket, please provide an explicit line in your description that you are giving Shapr3D consent to look at data related to your submitted file.

Verify your Shapr3D account

To fully access the Shapr3D Customer Support portal, you need to verify your Shapr3D account email address.

Verify your account

Verify your email address and complete your account registration through your account page:

1. Log in to your account page.
2. Select your email address at the top right corner.
3. In the drop-down menu, look for the alert about your unverified email address.
4. Select **Resend verification email** in the drop-down menu to receive an email from Shapr3D to complete your account registration.
5. Open the email and select **Verify email address**.

Verification issues

No email received

If you haven't received the verification email, try any of the following:

- **Check your registered email address for Shapr3D**: Go to your Account page to confirm the email for your Shapr3D account and ensure you're signed in to the correct email address.
- **Check your folders**: If you're organizing your emails using folders, the verification email may have ended up in one of your folders. Make sure to check all your folders including **Spam**.
- **Search for @shapr3d.com**: An easy way to find an email is through your email search bar. Simply type **@shapr3d.com** and look for the results in all your folders including **Spam**.
- **Check your spam filters**: Your email might be detecting the email as spam. Check your filters and make sure **@shapr3d.com** is not on the list.

Email detected as spam

If you find the email in your spam folder, follow these steps:

1. Move the email to your primary folder.
   Note: If the email remains in the spam folder, the verify email link will not work.
2. Click the **Verify email** link to verify your email.

Whitelist Shapr3D

In some cases, you may need to whitelist Shapr3D's domain or IP addresses if your school or company implements stringent email security measures. Reach out to your domain provider and request to whitelist the domain or IP addresses of Shapr3D:

- **Domain**: @shapr3d.com
- **IP addresses**:
    - 149.72.175.150
    - 149.72.227.206

Need more help?

If your email address is still unverified after you've completed the steps above, contact Shapr3D Support at **loginsupport@shapr3d.com** detailing your issue, and a team member will assist you promptly.

Create screen recordings for your request

When submitting a request to our support team, a screen recording is helpful in collecting information about your issue. To ensure that the format of screen recordings you submit is supported by Shapr3D, follow the tips below on how to share regular and larger files.

> **Important**
>
> If you choose to provide attachments of your work in your support ticket, please provide an explicit line in your description that you are giving Shapr3D consent to look at data related to your submitted file.

How to create screen recordings

Windows

Try these tips on how to screen record on Windows:

- Follow this guide on how to record your screen on Windows 11.
- You can also use the free Microsoft app, Screen Recorder Pro.

iPadOS

To learn how to screen record on your iPad, refer to this Apple support page: Record the screen on your iPhone, iPad, or iPod touch.

macOS

To learn how to screen record in a Mac device, refer to this Apple support page: Take screenshots or screen recordings on Mac.

visionOS

On Apple Vision Pro, you can capture your view or record your session using built-in gestures and controls. For detailed steps on how to take captures or recordings in visionOS, see Apple's guide: Take a capture or recording of your view in visionOS.

Find hardware and software specifications

When reaching out to Shapr3D support for technical issues, make sure to provide accurate hardware and software specifications. This information helps our team diagnose and resolve issues more efficiently. Follow the steps below to find the necessary details for your device.

Windows

Before submitting a support ticket for technical issues, make sure to provide all information such as CPU and GPU details from the **DirectX Diagnostic Tool.**

To get the necessary specifications for Windows:

1. Go to the **DirectX Diagnostic Tool**
2. Click **Save All Information** to export the necessary specifications.

For more detailed instructions, visit How to Extract Information from the DirectX* Diagnostic Tool.

iPadOS

Before submitting a support ticket for technical issues, make sure to provide the following specifications of your iPadOS device:

- iPad Model Name
- iPadOS Version
- Model Name
- Model Number

To get the necessary specifications for iPadOS:

1. Go to **Settings** > **General** > **About.**
2. Take a screenshot of the necessary specifications to include in your support ticket.

For more detailed instructions, visit Get information about your iPad.

macOS

Before submitting a support ticket for technical issues, make sure to provide the following specifications of your macOS device:

- Mac Model Name
- Graphics Card (GPU)
- macOS Version
- Memory (RAM)
- Processor (CPU)

To get the necessary specifications for macOS:

1. Go to the **Apple menu** > **About This Mac**.
2. Take a screenshot of the necessary specifications to include in your support ticket.

For more detailed instructions, visit Get system information about your Mac.

Educational license email whitelist guide

Shapr3D offers email whitelisting for accredited educational institutions. Whitelisting email domains give automatic approval for students, teachers, or faculty members who apply for the Shapr3D Educational license.

Requirements

Before email whitelisting can be requested, gather the following requirements:

- Signed and stamped document from the educational institution with the following information:
  - Must confirm that the students are currently associated with the institution
  - Must state that the accounts will be used for educational purposes only
- Complete list of the email domains used by the members of your institution

Email whitelisting process

For the representative making the request

To request for email whitelisting for your institution, follow these steps:

1. Contact Shapr3D by submitting a request to the support team. Include and attach the requirements you gathered in the request form.
2. Wait for a response from Shapr3D confirming that the request is approved and the email domains whitelisted.
3. Inform the members of your institution that the email domains are now whitelisted.

For the members applying for the Educational license

Follow these steps to get an Educational license automatic approval:

1. Make sure that your institution email domain is already whitelisted.
2. Create a Shapr3D account using the email address provided by your institution.
3. Verify your account.
4. Apply for an Educational license through the Request Educational license form.
   Note: If your email domain is already whitelisted, you don't need to attach any documents.
5. Wait up to 1 working day for your request to be automatically approved.

Troubleshooting

Installation issues

If you're having issues installing Shapr3D, try the suggested steps below to solve your issue.

Unable to install from the Microsoft Store

You can download the standalone installer through the Shapr3D website. For more details, visit Download Shapr3D.

If you prefer to install Shapr3D from the Microsoft Store, try these tips:

- **Make sure you're signed in to the Microsoft Store with your Microsoft account.**
  If you're not, select the user icon at the top right to sign in.
  Note*: Your Microsoft Store account might not be the same account you use to log in to your Windows device.* If you don't remember your password, visit Reset a forgotten Microsoft account password for tips on password recovery.
- **Check if your Windows version is compatible with Shapr3D.**
  Your device should be running a version of Windows 10, version 2004 or later. Refer to Which version of Windows operating system am I running? on how to check the OS version and Update Windows to update it.
- **Use a different Microsoft Store if you have the Educational or Business license for Shapr3D.**
  If your Microsoft account is connected to an educational institution, try installing Shapr3D from the Microsoft Store for Education.
  If your Microsoft account is a business account, try the Microsoft Store for Business.
- **Clear the Microsoft Store cache.**
  You can run an application, wsreset.exe, as the administrator. For more details, visit How to Repair or Reset Microsoft Store or Clear Windows Store Cache in Windows 11/10.
- **Check any error codes in Microsoft Store.**
  If the Microsoft Store shows an error code, such as "0x803F6101," check the meaning of the code in Microsoft Docs for store operations error codes and common installation-related error codes to help give you hints on what to do.

Multiple installation issues on Windows

Installing Shapr3D from multiple installation sources is not supported as it can cause data loss. If you get a message about multiple installations when opening Shapr3D, make sure that you only have one Shapr3D installed on your device.

Follow these steps to solve the issue:

1. From the Windows Start menu, open **Settings**.
2. Select **Apps > Apps & features**.
3. Sort the list by date.
4. Look for **Shapr3D**.
5. If you see 2 builds of Shapr3D, uninstall the most recently installed one.

For more details about uninstalling programs from Windows, visit Uninstall or remove apps and programs in Windows.

Shapr3D stability issues

Running Shapr3D on the latest drivers gives you the best modeling experience. If you experience stability issues while using Shapr3D, make sure your device software or drivers are up-to-date.

Symptoms

Below are the possible issues you may encounter if your device software or drivers are outdated:

- Shapr3D frequently crashing
- Unusual graphical glitches
- Problems launching Shapr3D

Troubleshooting

Windows

Try these steps to solve the issue:

1. Check that Shapr3D is up-to-date and restart your device. If the symptoms persist, go to the next step.
2. Identify the manufacturer of your GPU.
3. Choose your GPU manufacturer to learn how to update your drivers:
   - AMD
   - Intel
   - NVIDIA

   Note: Some device manufacturers provide custom drivers for their devices. Visit the website of your manufacturer to learn how to update your drivers.
4. Select the driver component you want to update.
5. Download and install the latest version of the driver.
6. Restart your device.
7. If you continue to experience issues, contact our Customer Support team.

To learn more about updating Windows drivers, visit Automatically get recommended drivers and updates for your hardware and Update drivers manually in Windows.

iPadOS

Try these steps to solve the issue:

1. Check that Shapr3D is up-to-date and restart your device. If the symptoms persist, go to the next step.
2. Check that your device software is up-to-date.
3. If you continue to experience issues, contact our Customer Support team.

macOS

Try these steps to solve the issue:

1. Check that Shapr3D is up-to-date and restart your device. If the symptoms persist, go to the next step.
2. Check that your device software is up-to-date.
3. If you continue to experience issues, contact our Customer Support team.

Third-party known issues

iOS and iPadOS sharing issues

There's a known issue in iOS 13 and later that prevents users from using AirDrop and other share options such as email to share their files with other devices and apps.

Symptoms

In reported cases, the **Save This File** window is blank and doesn't display any sharing options. This may prevent you from sharing your Shapr3D projects.

Workaround

Try any of the following workarounds to help solve the issue:

- **Reset network settings**: Go to **Settings** > **General** > **Transfer or Reset iPad** > **Reset** > **Reset Network Settings**.
- **Turn off Siri Suggestions**: Go to **Settings** > **Siri & Search** > Turn off **Show When Sharing** and then restart your iPad.

If the issue persists, visit Fix Issue of 'Sharing Options Not Showing' on iOS 16/15 for other workarounds you can try.

Error messages in History

When reviewing the History sidebar, you may occasionally notice yellow warning symbols beside certain steps. Yellow warning messages may appear for bodies or faces that can't be created due to missing or invalid results for certain steps with incorrect parameters or references. Read on to learn about the different History error messages and how to solve them.

Missing references

A common error you may encounter is a "missing reference" for affected History steps. These missing references can either be for sketch planes, profiles, or edges.

Plane

To resolve a missing reference for a sketch plane:

1. Expand the History card for the step with a missing plane reference.
2. Select **Fix**..
3. Select a planar face or plane.
4. To complete the fix, select **Done**.

Edge

To resolve a missing reference for an edge:

1. Expand the History card for the sketch plane with a missing edge reference.
2. Select **Fix**..
3. Select valid edges on your model.
4. To complete the fix, select **Done**.

Profile

To resolve a missing reference for a profile:

1. Expand the History card for the step with a missing profile reference.
2. Select **Fix**..
3. Select a valid profile.
   Note: The missing references are usually highlighted in yellow.
4. To complete the fix, select **Done**.

Fix broken dependencies in Design History

When you model in Shapr3D, each step that gets recorded in History forms a dependency or connection. If a step – such as an edge, sketch, or complete body – is missing, then the subsequent steps can't properly proceed, eventually breaking the history.

Common causes

If you have missing modeling elements or broken design history, it could be due to any of the following reasons:

- A reference geometry or step was lost or removed
- A reference or step was deleted

Symptoms

Your design history may have broken dependencies or connections if you experience any of the following:

- Missing modeling elements
- Entire model completely gone or missing
- Several error messages in History

Troubleshooting

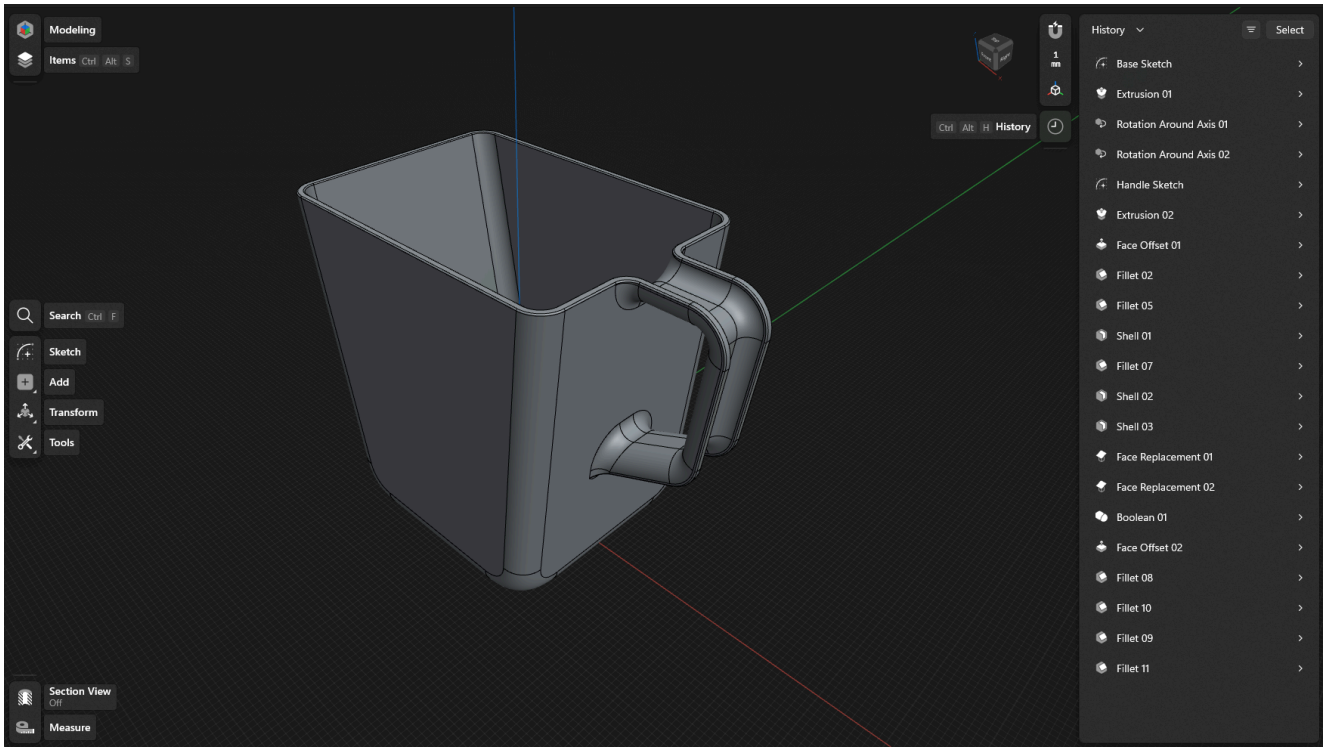
In most cases, you can fix broken connections or dependencies in your model by following these steps:

1. From the **History** sidebar, locate the first step with the first error message and then open the expandable card.
2. Select **Fix…** to see what reference is missing. Missing references are shown as an orange hatch in the modeling space.
3. Proceed with any of the following steps:
    - Recreate the missing reference:
        1. Create the missing reference in the same place as where the orange hatch was shown.
        2. **Optional:** If your newly created reference generates a new step in the History sidebar, move the newly created step above the step you're currently fixing.
    - Select a different reference:
        1. Choose a different reference for the broken step.
        2. To finish, select **Done**.

> **Tip**
>
> When encountering multiple error messages in Design History, start fixing the first step with the first error message.

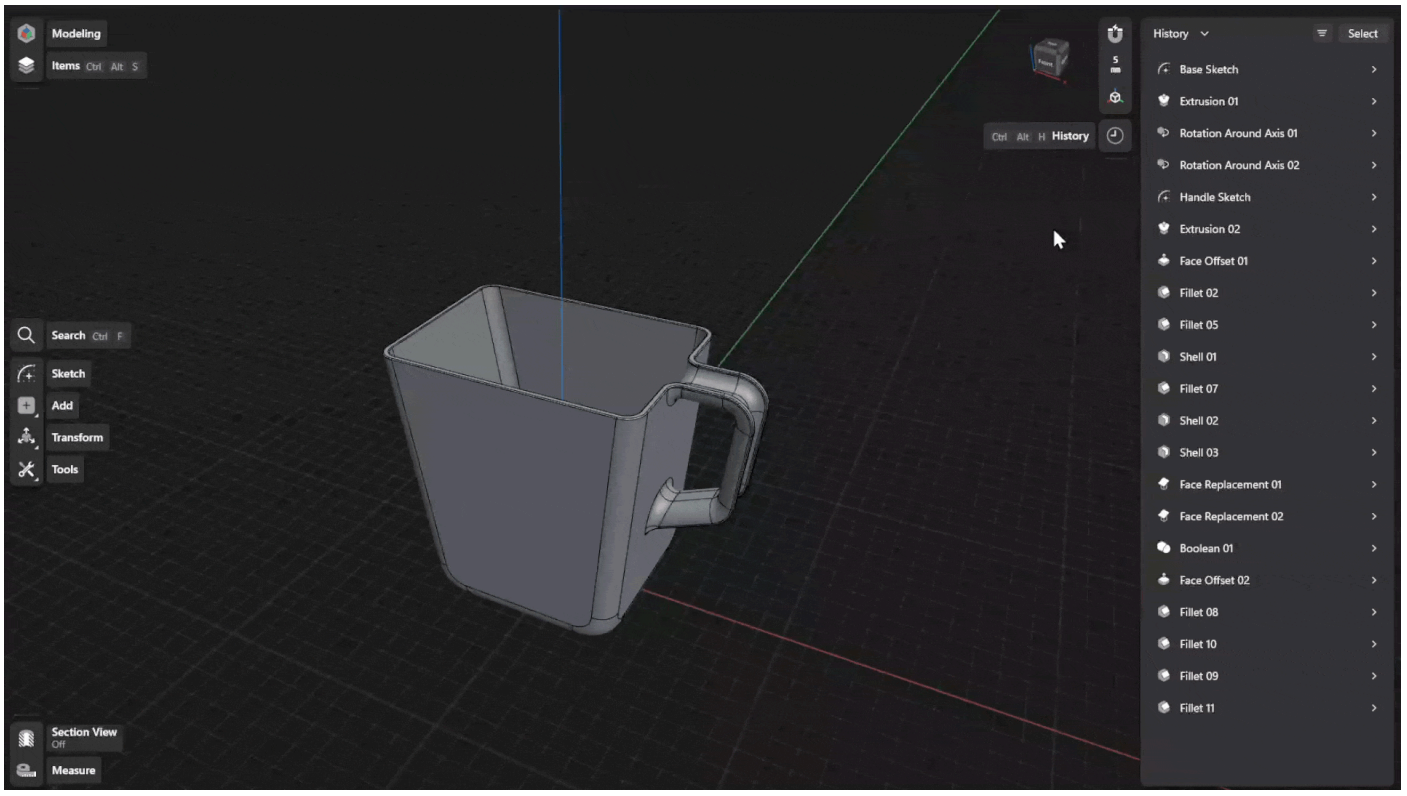
Example #1: Missing model due to a deleted reference step

In the example below, we have a model with a properly working Design History.



Scenario

In this scenario, we will delete the **Base Sketch** step at the very beginning of the History sidebar.



By removing the **Base Sketch** step, we eliminate the reference of all the succeeding steps, causing each step in the design history to break, and the model to disappear as shown by the yellow warning labels.

Troubleshooting

To fix this issue:

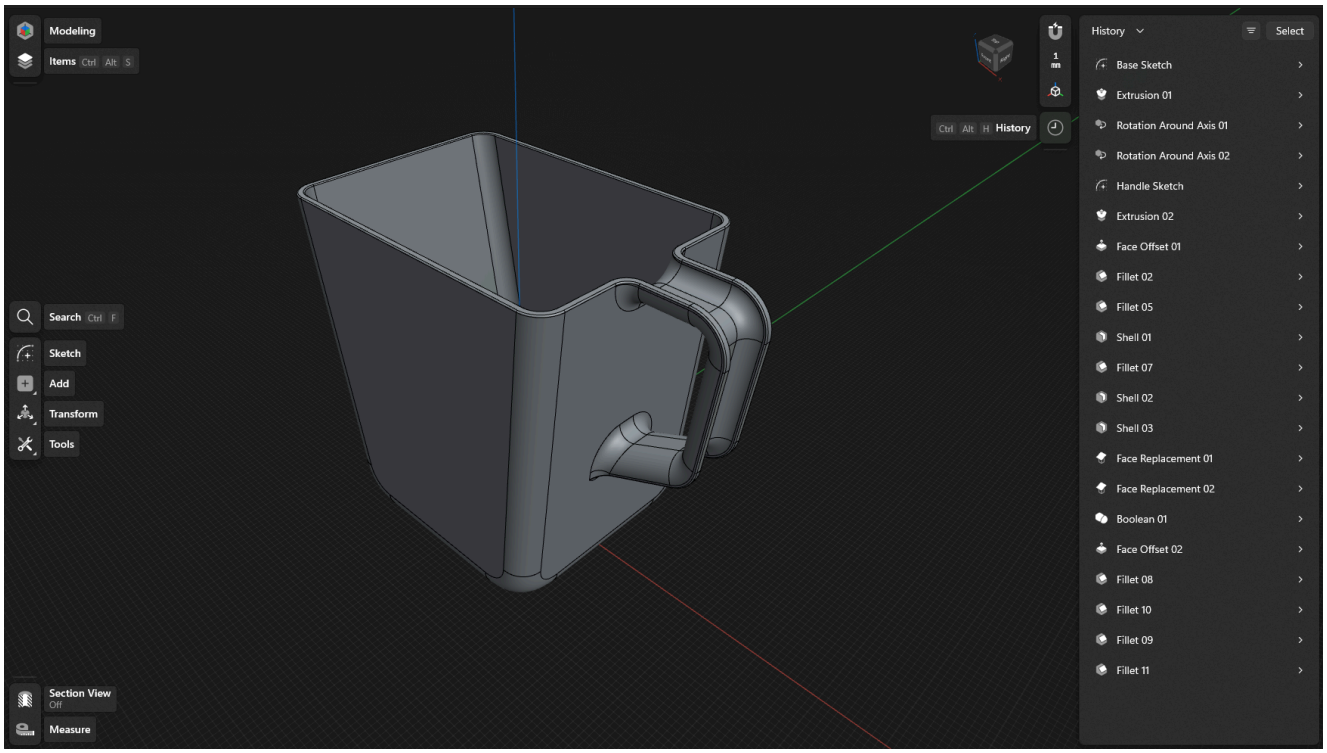
1. Open the expandable card for the first step with the first error message. In this case, we expanded **Extrusion 01**.
2. In the card, check the the description of the error message to see what's causing the problem. In this case, it states that we have 1 missing reference.
3. To view the missing reference, click the **Fix...** button. Selecting **Fix...**highlights the missing cross-section in the modeling space with an orange hatch, which was from where the deleted **Base Sketch** step was originally created.
4. We can then either use the orange highlight to recreate the cross section, or select another sketch that will be used as the new reference.

Tip

Design History maintains a connection between bodies and sketches. If a sketch is in the way, you can use the **Items Manager** to hide it instead of deleting it. Hidden sketches will not be visible in the modeling space and will not affect the performance of Shapr3D.

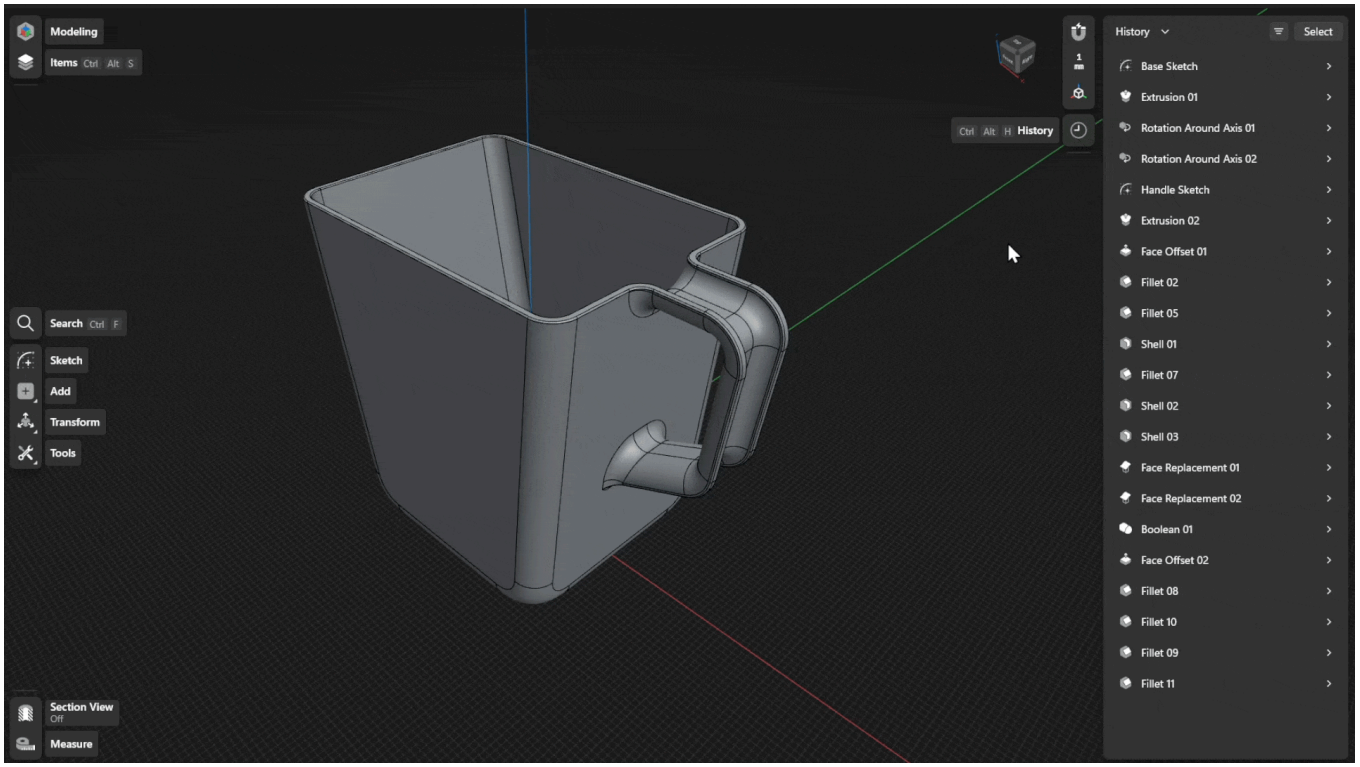
Example #2: Missing model parts due to a deleted sketch item

In the example below, we have the same model with a properly working Design History.



Scenario

In this scenario, we will delete a sketch item that is part of the **Handle Sketch** step.



By deleting a sketch item that is part of the **Handle Sketch** step, the sketch handle is no longer a closed cross section that can be used as a reference, causing the succeeding relevant steps to break, and the handle of the model to disappear as shown by the yellow warning labels.

Note: The steps **Shell 02** and **Fillet 09** did not break because they were steps for the main container so they were not affected by the change.

Troubleshooting

To fix this issue:

1. Open the expandable card for the first step with the first error message. In this case, we expanded **Extrusion 02**.
2. In the card, check the description of the error message to see what's causing the problem. In this case, it states that we have 1 missing profile reference.
3. To view the location of the missing reference, click the **Fix...** button. Selecting **Fix...** highlights the missing cross-section in the modeling space with an orange hatch, in this case, due to the deleted sketch item.
4. We can then either use the orange highlight as a reference so we know where to recreate the incomplete cross section.
5. **Optional**: If the sketch is automatically hidden, we can go to the **Items Manager** to show the recreated sketch for the **Sketch Handle** step.
6. To finish the fix, we can now go back to the **Extrusion 02** step to add the missing profile.
7. After selecting **Done**, all the broken steps are now repaired and the full model now appears again in the modeling space.

Rendering accuracy

In Shapr3D, rendering accuracy may differ from modeling accuracy. While Shapr3D uses a Parasolid modeling engine to create manufacturable solid bodies with exceptional precision, this precision doesn't necessarily apply to the display accuracy.

Depending on the hardware you use to run Shapr3D, you may notice a few display inaccuracies such as:

- gaps between sketches and their fillings
- gaps between bodies and their sketches
- jagged edges

These display inaccuracies you notice don't affect the precision of your model.

Read on to learn more about the concepts behind modeling and rendering accuracy, and some troubleshooting tips to get the best display accuracy when you're modeling.

Modeling vs rendering precision

In CAD, modeling is the process of creating precise geometrical elements while rendering is the general term for displaying a precisely computed geometrical element.

Modeling precision

When you model, geometric calculations of created sketches and models such as boundaries, intersecting lines, intersecting bodies, or super complex splines, are carried out with a precision of at least 0.001 mm in the background.

The precision of geometric calculations in the background is easy to maintain for modeling, but this precision isn’t always easily carried out in rendering.

Rendering precision

To ensure the efficiency of the required hardware performance and rendering, the level of precision maintained in model calculations goes through a geometric simplification which causes you to only see an approximation of your model.

Complex models accuracy

Your complex models are still geometrically accurate regardless of reduced rendering quality.

The reduced rendering accuracy is just a visibility setting to ensure the efficiency of the required hardware performance and rendering.

With a reduced rendering quality, Shapr3D saves performance and can run flawlessly on a broad range of devices.

Troubleshooting rendering quality issues

You might experience reduced rendering quality more frequently when you zoom in a lot in the modeling space or have hundreds of bodies.

The scenarios below give you an overview of how zooming affects the rendering quality of the different areas in your model and what you can do to make the quality more accurate.

Reduced rendering accuracy on sketches

Cause

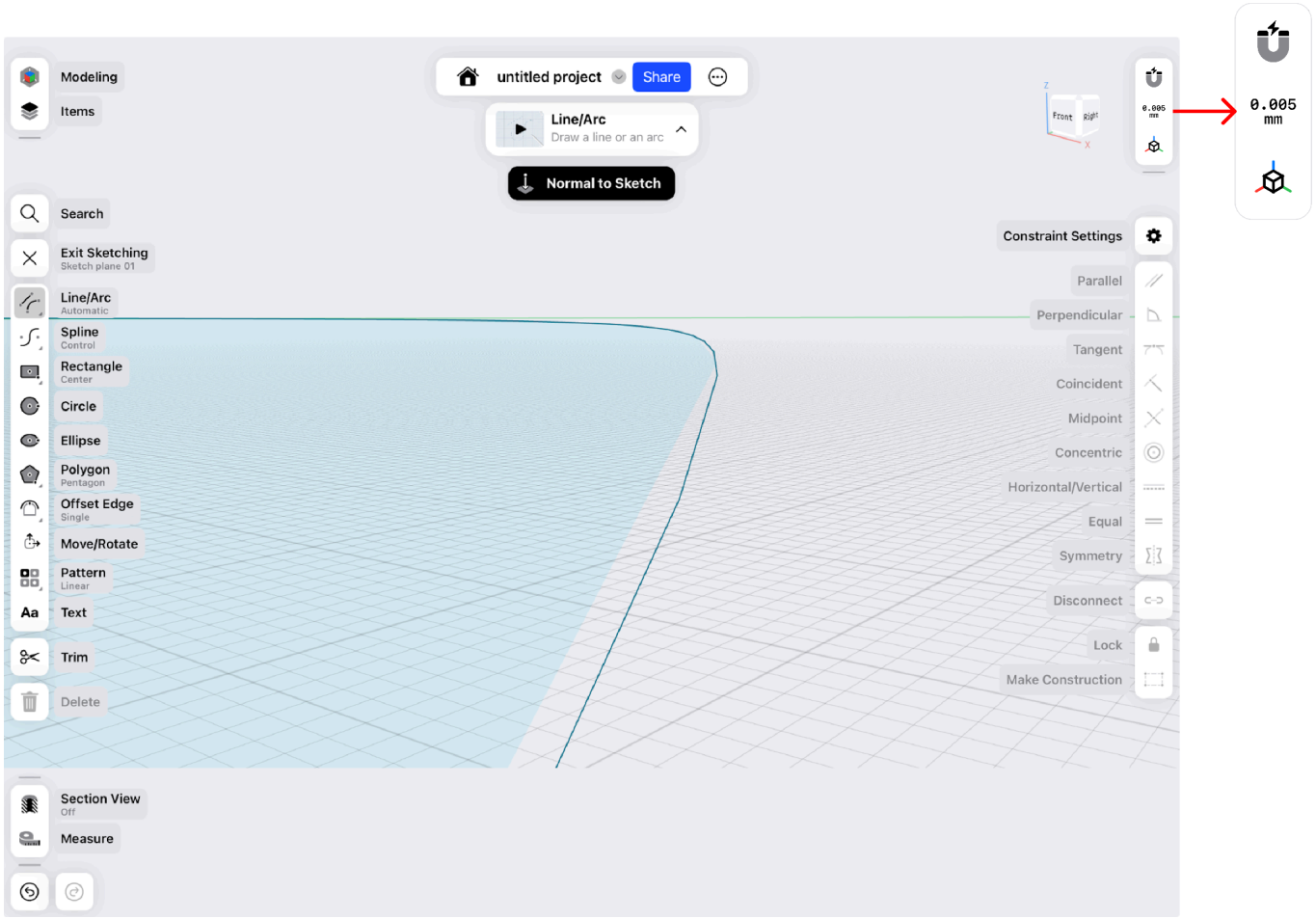
When zooming in on sketches, Shapr3D reduces the rendering quality to save on performance.

Symptoms

With the reduced quality, you may experience the following:

- Visible gaps between sketch elements and the sketch profile
- The camera intersects with the grid

In the example below the workspace is zoomed in with a grid resolution of 0.005 mm. The level of zoom shows an intersection between the camera and the grid and a gap between the sketch and the blue sketch profile.



Troubleshooting

Try these steps to improve the rendering accuracy:

1. Check the grid resolution in the upper right corner, and zoom out to adjust the resolution accordingly.
2. Carefully zoom out to get the right level of detail, or double-tap the Orientation Cube to give you a clear overview of the entire model.

Distance between bodies and sketches

Cause

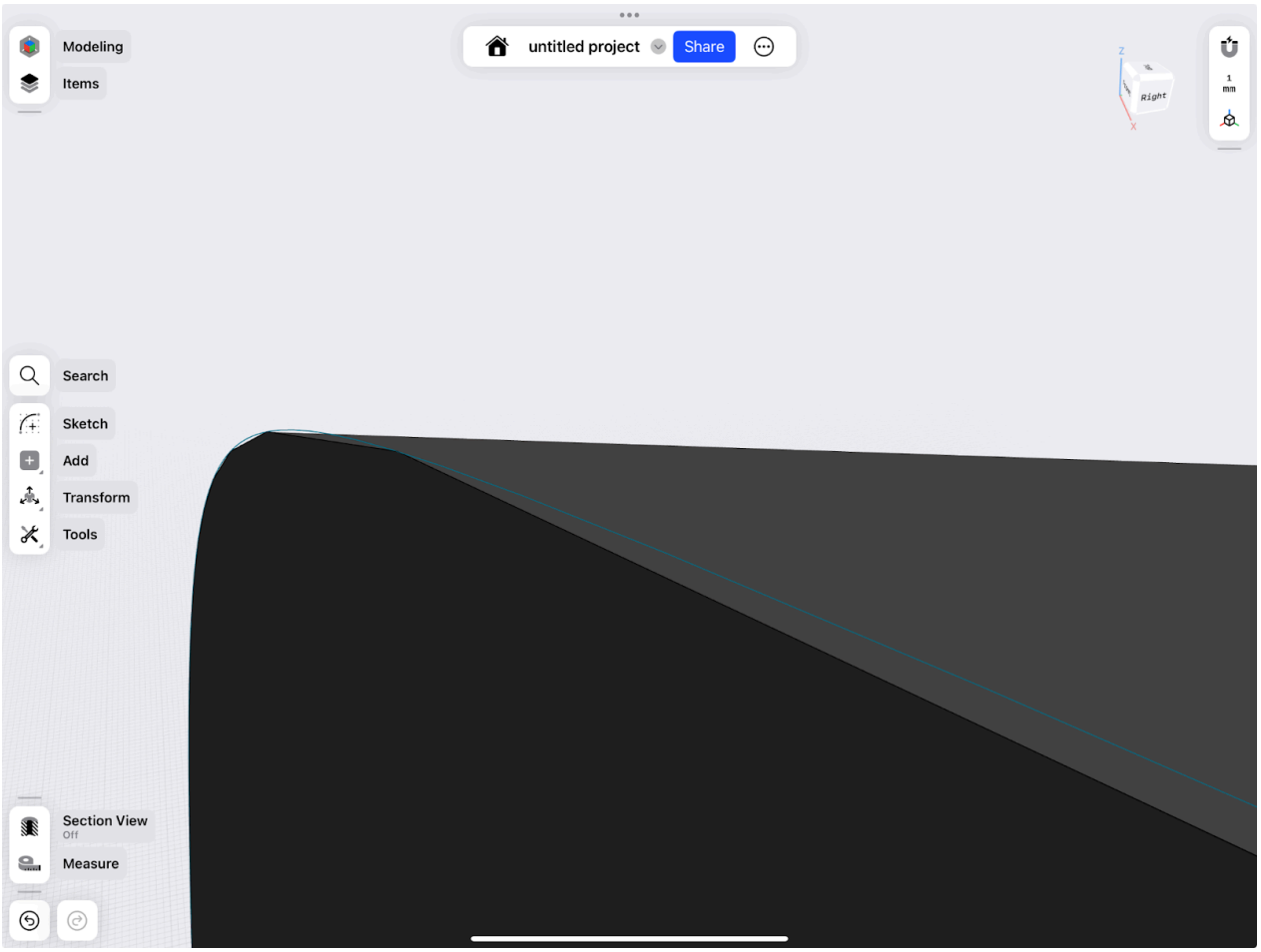
When zooming into the edges of bodies, Shapr3D reduces the rendering quality to save on performance. The reduced rendering quality doesn't affect the computational accuracy of the model.

Symptoms

With the reduced quality, you may experience the following:

- Visible gap between the edge of a body and the sketch from which it was created

In the example below, there is a visible gap between the sketch and the edge of the body.



Troubleshooting

To improve the rendering accuracy, adjust the level of zoom to achieve your level of desired rendering accuracy.

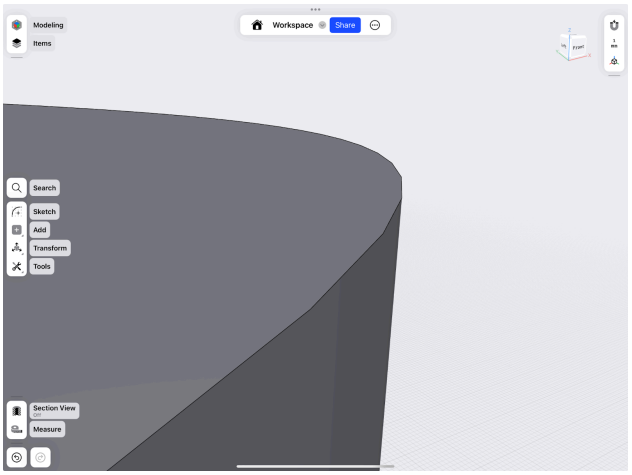
Curved edges appearing as jagged edges

Cause

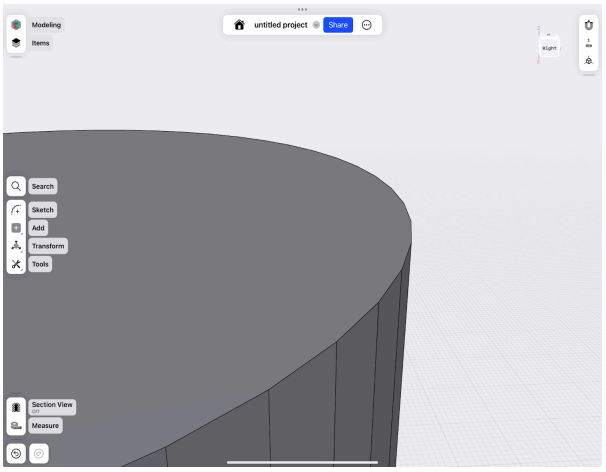
- When you have A few hundred bodies in your modeling space and you zoom in on edges, Shapr3D reduces the rendering quality to provide a smooth modeling experience.
- To save on performance, the detail of rendering may decrease depending on your level of zoom and the number of bodies in the modeling space. The reduced rendering quality doesn't affect the computational accuracy of the model.
- Zooming in on faceted models consisting of polygons or straight elements eventually appear as jagged edges when zoomed in at a certain level.

In the examples below, the model on the left shows a jagged but continuous edge, showing that it is a curved edge.

The model on the right shows intersecting edges indicating that the upper edge is not a curve but an edge consisting of straight elements.



Curved edge with reduced rendering quality



Edge consisting of multiple straight elements

Symptoms

With the reduced quality, you may experience the following:

- Curved edges appear jagged the more you zoom
- Consider using splines or arcs to achieve smoother edges

Troubleshooting

To improve the rendering accuracy, adjust the level of zoom to achieve your level of desired rendering accuracy.

If the troubleshooting steps above didn't improve the rendering accuracy of your model, try these additional tips:

- Make sure that the drivers on your device are up-to-date. To learn more, visit Shapr3D stability issues.
- Create your models near the origin. Check out Move/Rotate (3D) and Move/Rotate (Sketch) to learn how to move your models near the origin.

If the issue persists, submit a request.

Password reset issues

[Article link | Table of contents | Back]

If you're having trouble resetting your Shapr3D account password or experiencing issues such as not receiving the password reset email, try the solutions below to help resolve your issue.

No email received

If you haven't received the password reset email, try any of the following:

- **Check your registered email address for Shapr3D:** Ensure you enter the correct email address associated with your Shapr3D account for the password request. Verify that there are no typos or unnecessary punctuation.
- **Check your folders**: If you're organizing your emails using folders, the password reset email may have ended up in one of your folders. Make sure to check all your folders including **Spam**.
- **Search for @shapr3d.com**: An easy way to find an email is through your email search bar. Simply type **@shapr3d.com** and look for the results in all your folders including **Spam**.
- **Check your spam filters**: Your email might be detecting the email as spam. Check your filters and make sure **@shapr3d.com** is not on the list.

Email detected as spam

If you find the email in your spam folder, follow these steps:

1. Move the email to your primary folder.
   Note: If the email remains in the spam folder, the password reset link will not work.
2. Click the **Reset password** link to change your password.
   Note: For the complete steps, visit Reset your Shapr3D account password.

Whitelist Shapr3D

In some cases, you may need to whitelist Shapr3D's domain or IP addresses if your school or company implements stringent email security measures. Reach out to your domain provider and request to whitelist the domain or IP addresses of Shapr3D:

- **Domain**: @shapr3d.com
- **IP addresses**:
  - 149.72.175.150
  - 149.72.227.206

Need more help?

If you need further assistance, contact Shapr3D support at **loginsupport@shapr3d.com**. For a quicker resolution, include the following information in your email:

- Any other email addresses that you might have an account with
- The date and amount of your latest subscription payment
- A screenshot of the charge or a section of your bank statement displaying the charge

Licenses

Third-party licenses

[Article link | Table of contents]

Shapr3D uses several third-party libraries, repositories, and other similar software packages. Thank you to the owners and contributors listed below.

- 3Dconnexion
- abseil
- Alamofire
- AppsFlyerFramework
- Dear ImGui
- Draco 3D Graphics Compression
- Eigen
- FBSDKCoreKit
- Filament
- Firebase/Core
- Firebase/Crashlytics
- Intel TBB
- Lib3mf
- LLVM
- Meshoptimizer
- MGSwipeTableCell
- MKStoreKit
- MikkTSpace
- Miniz
- moodycamel::ConcurrentQueue
- moodycamel::ReaderWriterQueue
- mpack
- pugixml
- RapidJSON
- Reachability
- spdlog
- SQLite
- stb_image
- SwiftLint
- tinyobjloader
- TL::expected
- tree-sitter
- USD
- Welder
- Xatlas