

 **shapr3D** | **MANUAL**

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Getting started with Shapr3D

Requirements, downloading, subscriptions, accounts, storage, Shapr3D Cloud
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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 is available with a [monthly or yearly subscription](#). To prepare for software download and installation, start by checking the [compatibility page](#).

Access the download links here:

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

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

Windows

From the installer

Access the download link in the following ways:

- Choose your preferred installer here: shapr3d.com/download/windows, or choose directly from the following:
 - [Install directly \(Recommended\)](#): For systems that can connect directly to the internet. Using this option, Shapr3D and its dependencies are kept up-to-date automatically.
 - [Download package](#): For enterprise or restricted systems.

Important

This option requires you to download and install the latest version of [Visual C++ Redistributable](#). This method does not update Shapr3D automatically. Visit [our website](#) regularly to get the latest updates.

- Install via the command line: For enterprise or restricted systems. Download the [.appinstaller file](#) to your computer and then use the following command in a PowerShell console:
`Add-AppxPackage -AppInstaller "Shapr3D.Package.appinstaller"`
- 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

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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.

Windows

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 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 **Library**.
3. At the upper-right corner of the **Library** page, click **Get 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

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

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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 EDU 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

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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

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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**
 - You can also receive invitations to join other teams.
 - To become a team member of other teams, you need to have a **Shapr3D Website** subscription.
- **Collaboration**
 - With teams, you and your teammates can share projects in a single space.

Discounts and refunds

For App Store subscriptions, discounts from Shapr3D are not applicable and you can only request a refund from Apple if you decide to cancel your subscription.

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 account.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**.

The screenshot shows the Shapr3D pricing page. At the top, there is a navigation bar with the Shapr3D logo and menu items: PRODUCTS, DESIGN TEAMS, MANUFACTURING TEAMS, INDUSTRIES, and LEARN. On the right, there are links for CONTACT SALES and a blue Sign up button.

A single subscription covers Mac, iPad, and Windows.

Use the same account across platforms.

At the top of the pricing section, there are two tabs: Yearly (selected) and Monthly. A red badge above the Yearly tab says "Save 34%".

Basic	Pro	Enterprise
Modeling essentials	From concept to manufacture-ready models	Custom solutions for your team
\$0 /month	\$25 /seat/month <small>\$299 Billed annually</small>	Contact us for pricing
Including: <ul style="list-style-type: none">• Maximum of 2 projects• All 3D modeling tools• Import from major CAD platforms• Low-res export for prototyping• Shapr3D Cloud	Everything in Basic, plus: <ul style="list-style-type: none">• Teamspace for collaboration• 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	Everything in Pro, plus your choice of: <ul style="list-style-type: none">• Reassignable seats• Enterprise-compliant security & data management• Easy access with team login or single sign-on (SSO)• PLM integration• Consolidated billing• Customized training• Dedicated account manager
Select Basic	Start 14-day free trial <small>Free for Students and Educators</small>	Contact sales

All prices are in USD and exclude tax.

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.

shapr3D john.doe@domain.com ▾

Subscribe now
Access to all features without limits.

Number of seats (Including you)

- 4 +

Credit card details

1234 5678 9101 1121 12/24 •••

United States ▾ 62704

Billing information (Optional)

This information will be displayed on your invoice and used to calculate taxes.

Name on the bill

Address line 1

Address line 2 (Optional)

City

Country ▾ Post code

State / Region

Summary

Shapr3D Pro	\$1196
4 seat × \$299 × 1 year	
Tax (VAT)	\$0

Total due today \$1196

[Add coupon code](#)

Subscribe now

Your card will be charged when you click Subscribe now.

You will be charged \$1196 yearly unless you cancel your subscription or manage your team's seats.

6. Under **Credit card details**, enter your card payment details

shapr3D john.doe@domain.com ▾

Subscribe now
Access to all features without limits.

Number of seats (Including you)

- 4 +

Credit card details

1234 5678 9101 1121 12/24 •••

United States ▾ 62704

Billing information (Optional)

This information will be displayed on your invoice and used to calculate taxes.

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Summary

Shapr3D Pro	\$1196
4 seat × \$299 × 1 year	
Tax (VAT)	\$0

Total due today \$1196

[Add coupon code](#)

Subscribe now

Your card will be charged when you click Subscribe now.

You will be charged \$1196 yearly unless you cancel your subscription or manage your team's seats.

7. If required, enter your billing details under **Billing information**.

Subscribe now
Access to all features without limits.

<p>Number of seats</p> <p>– 4 +</p> <p>Credit card details</p> <p>1234 5678 9101 1121 12/24 ***</p> <p>United States 62704</p> <div style="border: 2px solid red; padding: 5px;"><p>Billing information (Optional)</p><p>This information will be displayed on your invoice and used to calculate taxes.</p><p>Name on the bill</p><p>Address line 1</p><p>Address line 2 (Optional)</p><p>City</p><p>Country Post code</p><p>State / Region</p><p>VAT number (HU12345678)</p></div>	<p>Summary</p> <table border="0"><tr><td>Shapr3D Pro</td><td style="text-align: right;">\$1200</td></tr><tr><td>4 seat × \$25 × 12 month</td><td></td></tr><tr><td>Tax (VAT)</td><td style="text-align: right;">\$0</td></tr><tr><td colspan="2"><hr/></td></tr><tr><td>Total due today</td><td style="text-align: right;">\$1200</td></tr></table> <p style="text-align: center;">Add coupon code</p> <p style="text-align: center;">Subscribe now</p> <p><small>Your card will be charged when you click Subscribe now.</small></p> <p><small>You will be charged \$1200 yearly unless you cancel your subscription or manage your team's seats.</small></p>	Shapr3D Pro	\$1200	4 seat × \$25 × 12 month		Tax (VAT)	\$0	<hr/>		Total due today	\$1200
Shapr3D Pro	\$1200										
4 seat × \$25 × 12 month											
Tax (VAT)	\$0										
<hr/>											
Total due today	\$1200										

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

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**.

shapr3D john.doe@domain.com

Subscribe now

Access to all features without limits.

Credit card details	Summary
<input type="text" value="Card Number"/> MM / YY	Shapr3D Pro Yearly Subscription €299
United States <input type="text" value="Post code"/>	Tax (VAT) €0
Billing information This information will be displayed on your invoice and used to calculate taxes.	Total due today €299
<input type="text" value="Name on the bill"/>	Add coupon code
<input type="text" value="Address Line 1"/>	<input type="button" value="Pay now"/>
<input type="text" value="Address Line 2 (Optional)"/>	
<input type="text" value="City"/>	
United States <input type="text" value="Post code"/>	
State <input type="text" value="Tax ID (Optional)"/>	

2. In the **Enter code** field, enter your coupon code, then select **Apply** to continue.

Credit card details	Summary
<input type="text" value="Card Number"/> MM / YY	Shapr3D Pro Yearly Subscription €299
United States <input type="text" value="Post code"/>	Tax (VAT) €0
Billing information This information will be displayed on your invoice and used to calculate taxes.	Due on January 29 €299
<input type="text" value="Name on the bill"/>	Total due today €0
<input type="text" value="Address Line 1"/>	<input type="text" value="Enter code"/> <input type="button" value="Apply"/>
<input type="text" value="Address Line 2 (Optional)"/>	<input type="button" value="Start Free Trial"/>
<input type="text" value="City"/>	Your trial ends on January 29. You will be charged €299 yearly unless you cancel your subscription.
United States <input type="text" value="Post code"/>	
State <input type="text" value="Tax ID (Optional)"/>	

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 account.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**.

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

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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 Shapr3D 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 EDU license

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A Shapr3D EDU license is a free version of Shapr3D — available in certain 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 EDU license is available for renewal as long as you're still part of an accredited educational institution.

Choose a topic to learn more:

- [EDU license eligibility](#)
- [Edu license FAQ](#)
- [Apply for an EDU license](#)
- [Renew your EDU license](#)

Important

Unlike Shapr3D Pro, a Shapr3D EDU license doesn't offer the following:

- Teams feature
- Priority technical support

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

EDU license eligibility

To be eligible for the Shapr3D EDU 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 EDU license](#).

Tip

If you've already started a Pro trial, [cancel the Pro subscription](#) before you apply for an EDU license.

EDU license availability FAQ

Due to a recent restructuring of the EDU 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 EDU license but I can see that it's not available in my country anymore. Why is that?

A: Due to some recent changes, we are currently restructuring our educational program. In the meantime, you can check our [pricing page](#) for other available options.

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

A: The relaunch has no definite time at the moment, but we will be sure to announce if anything changes. In the meantime, you can check our [pricing page](#) for other options available for you.

Q: I'm a teacher and I've been teaching Shapr3D for two years now. My EDU 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 EDU license from your website. Can you please help?

A: The availability of the EDU 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 EDU 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 EDU 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 EDU 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 EDU 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 EDU 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 EDU 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 EDU license

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

1. Make sure you [qualify for an EDU 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 EDU license** form.

Tip

If you're a faculty member who wants to request to whitelist a batch of email domains, visit [EDU 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 EDU 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 EDU 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 EDU license

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

When your EDU 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 EDU license or subscribe to [Shapr3D Pro](#), you'll regain access to all your projects immediately.

To renew your EDU license, simply [apply for an EDU license](#) again.

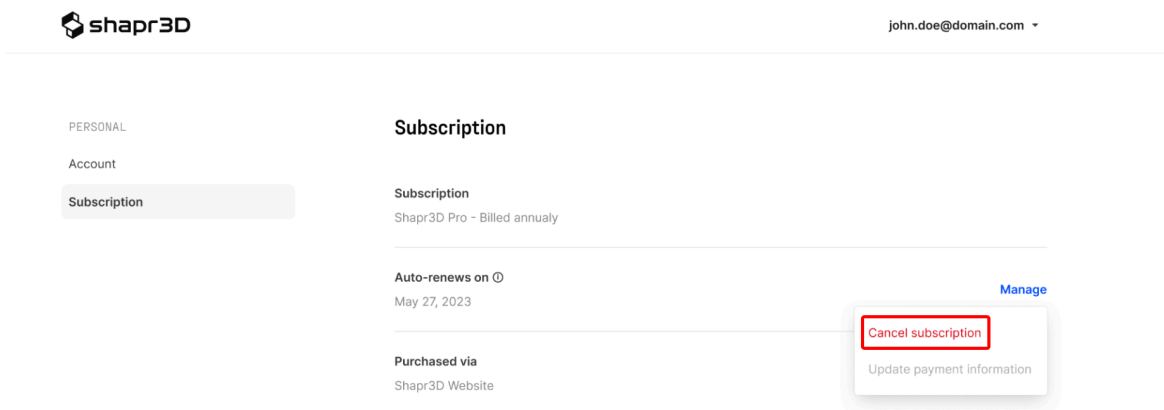
Cancellation and refunds

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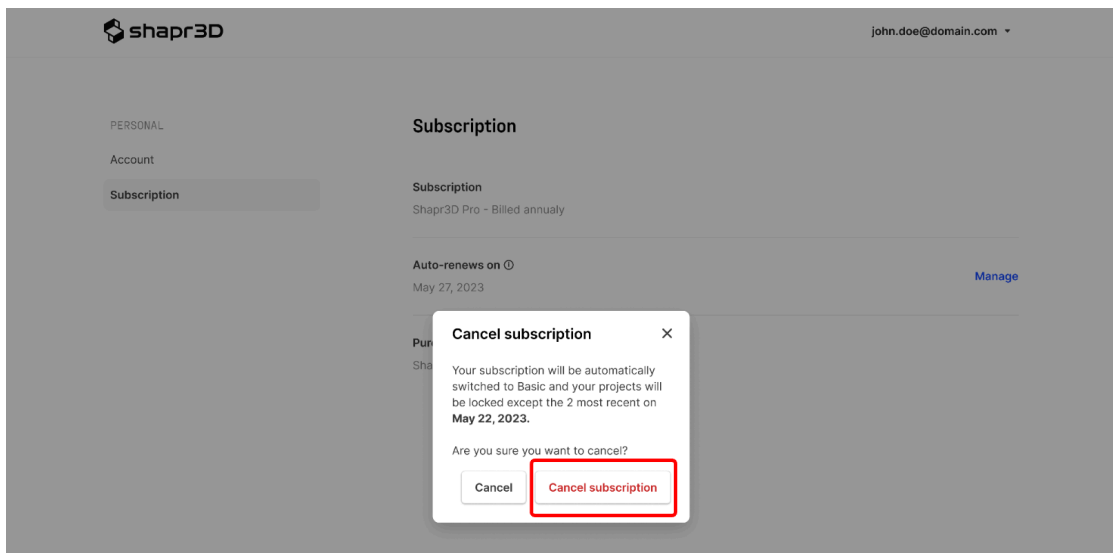
Depending on your method of subscription, you can cancel either from the **Shapr3D website** or the **App Store**.

Cancel via the Shapr3D website

1. Go to account.shapr3d.com and sign in.
Note: You can also log in by going to shapr3d.com > **Sign up**.
2. Go to **Subscription**.
3. In the **Auto-renews on** section, 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.

The screenshot shows the Shapr3D user interface. At the top left is the Shapr3D logo, and at the top right is the email address 'john.doe@domain.com'. On the left side, there is a navigation menu with 'PERSONAL' at the top, followed by 'Account' and 'Subscription' (which is highlighted). The main content area is titled 'Subscription' and shows 'Shapr3D Pro - Billed annually'. Below this, there is a section for 'Expires on' with a calendar icon and the date 'May 27, 2023'. A blue callout box contains the text 'Turn on auto-renewal to prevent losing access to Pro features at the end of the billing cycle.' and a button labeled 'Turn on auto-renewal' which is highlighted with a red border. At the bottom, there is a section for 'Purchased via' showing 'Shapr3D Website'.

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 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 account.shapr3d.com.
 - **From Shapr3D:** Open Shapr3D and then select **Log In**.
2. From the **Log In** page, select **Sign up**.
3. Fill out the required fields for **Work email**, **Password**, **Re-enter password**, and **Your name**.
4. Toggle the switch to accept the Terms of Use and Privacy Policy, then select **Sign Up**.
5. 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.

Reset your Shapr3D account password

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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 account.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 personal account and team settings

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You can manage your personal account and team settings on your Shapr3D account page. From the Shapr3D account page, you can:

- Update your email address and password
- Subscribe or view your subscription plan
- Manage your billing information
- Invite members to your team
- Manage your team settings
- View other teams you're a part of

Access your account page

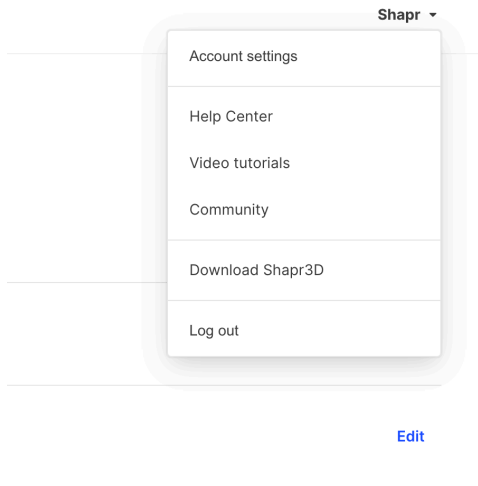
You can access your account page in the following ways:

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

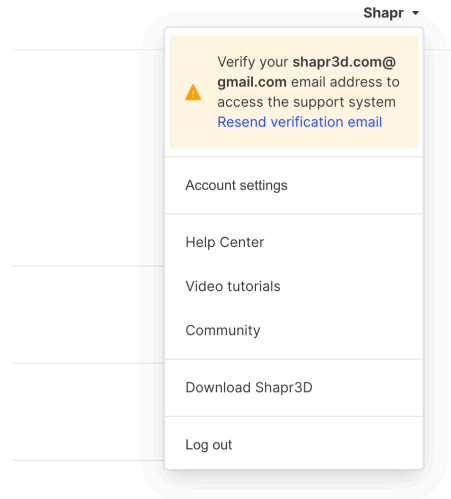
Profile

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



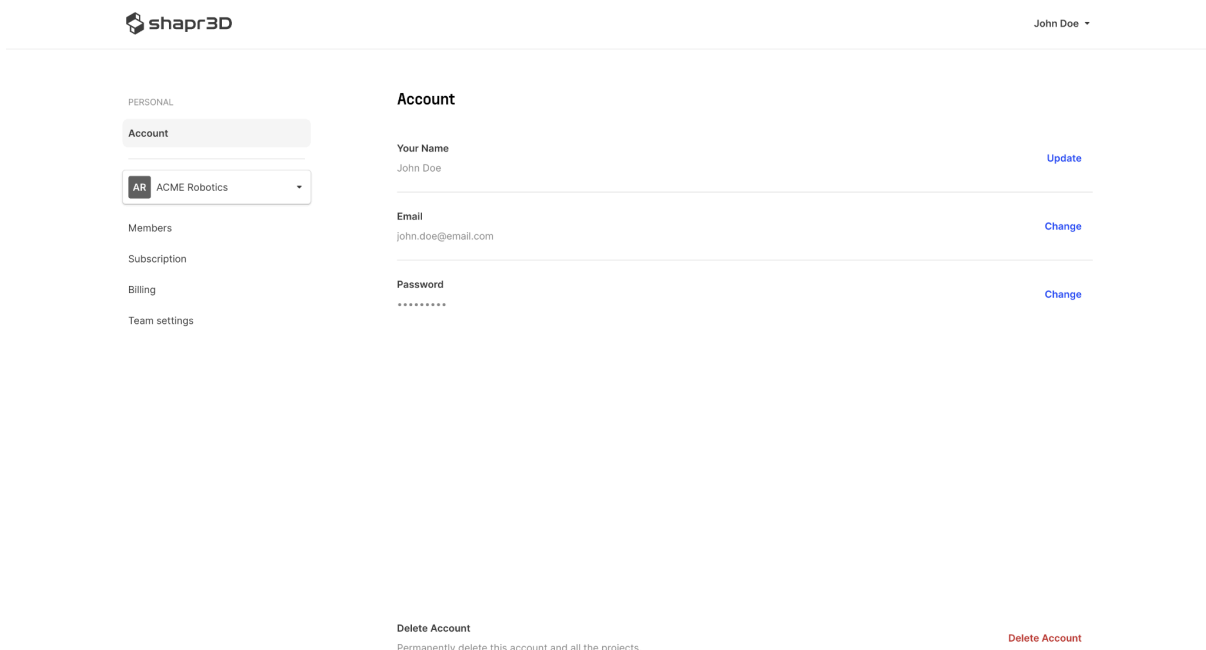
Unverified account

Manage personal account settings

To access your personal account settings, click/tap **Account settings**.

From the **Account** 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**.
- **Delete account** – To delete your account, select **Delete Account**. You can also access this option from Shapr3D by going to [Settings or Preferences](#).



Manage teams

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**
 - You can also receive invitations to join other teams.
 - To become a team member of other teams, you need to have a **Shapr3D Website** subscription.
- **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), EDU subscriptions, and App Store-based subscriptions.

Roles in teams

In teams, you can either be an owner or an editor.

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 Teamspace

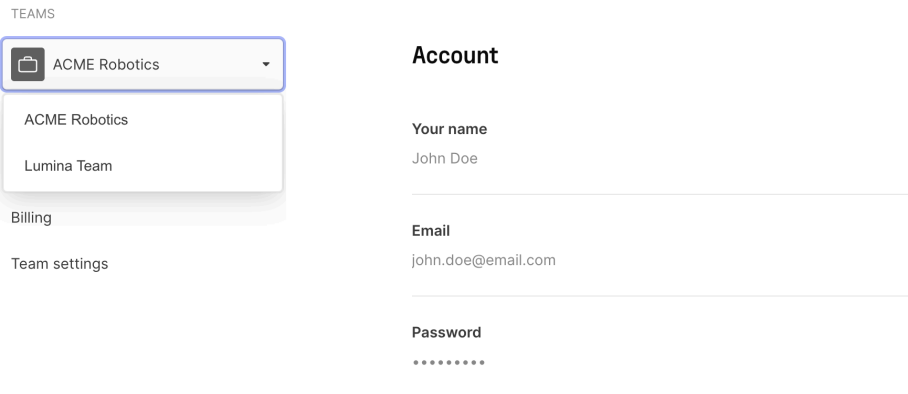
Manage team members as a team owner

Invite team members

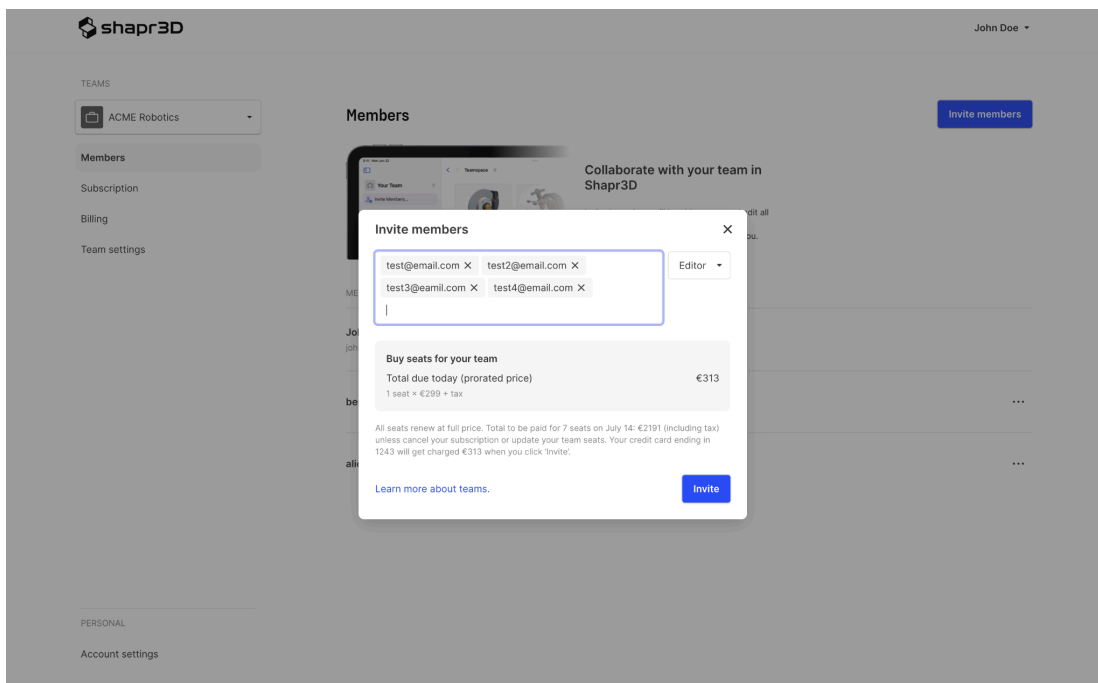
Team owners can add or remove members from the team directly.

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 editor access
- A team member occupies one purchased empty seat
- The team automatically gets charged for a new seat if all purchased seats are already assigned to a member

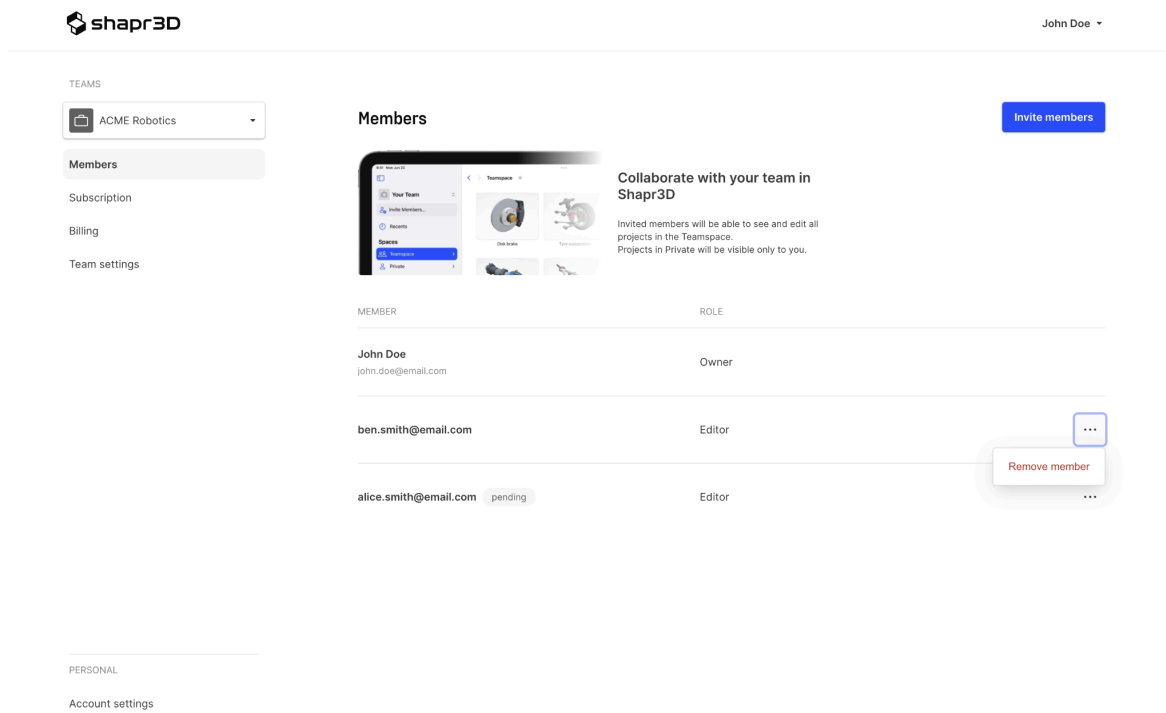
If members are removed from the team:

- 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

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

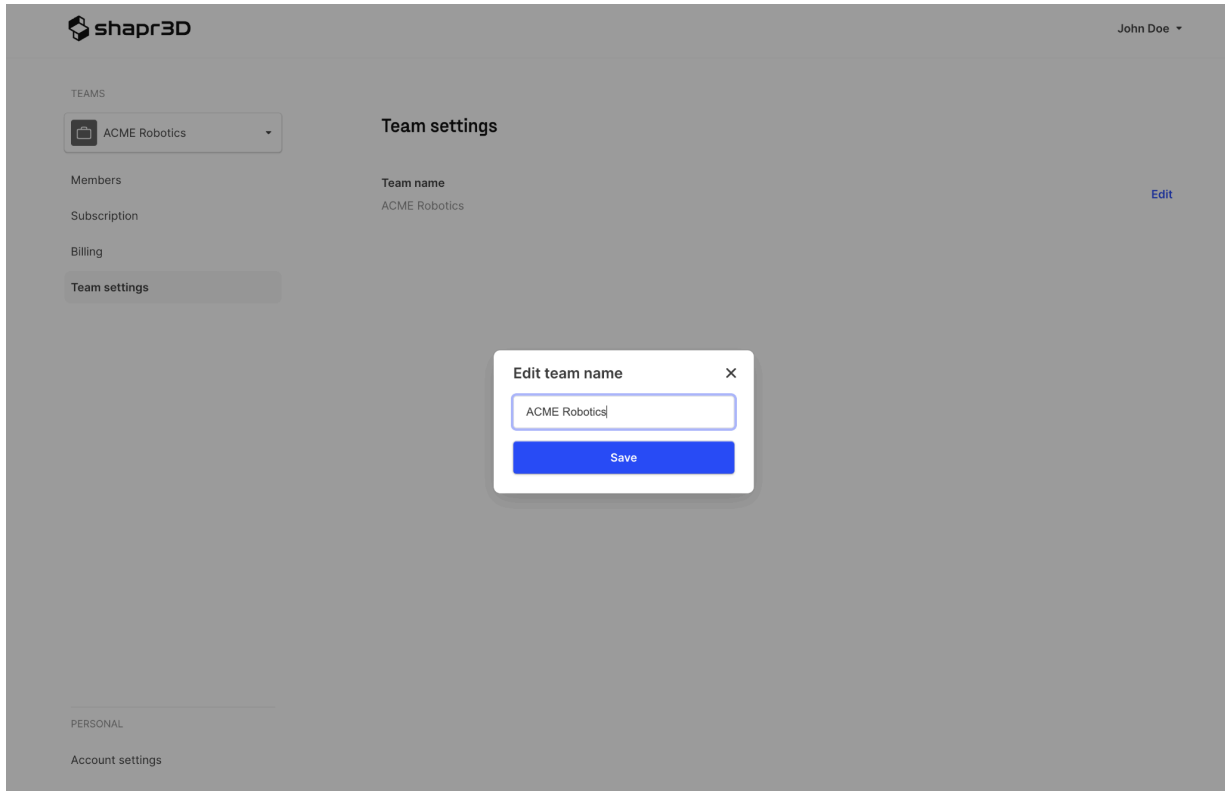
Your team subscription is handled by the team owner who sent you the invitation.

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**.

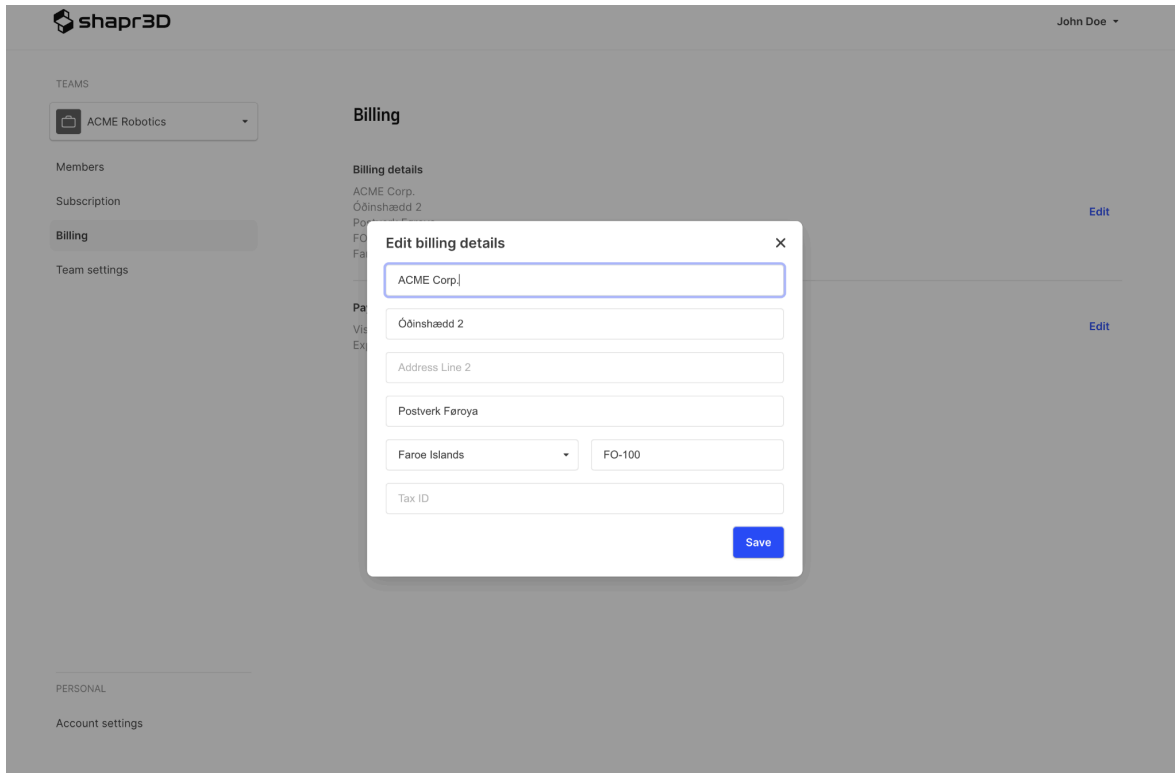


Billing

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**.



Data ownership

All the data and Intellectual Property (IP) created within a team belong to the owner.

Manage your 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**.



John Doe ▾

TEAMS

ACME Robotics ▾

Members

Subscription

Billing

Team settings

Subscription

Subscription

Shapr3D Pro - Billed annually

Seats

6

[Manage](#)

Auto-renews on

July 14, 2024

[Manage](#)

Purchased via

Shapr3D Website

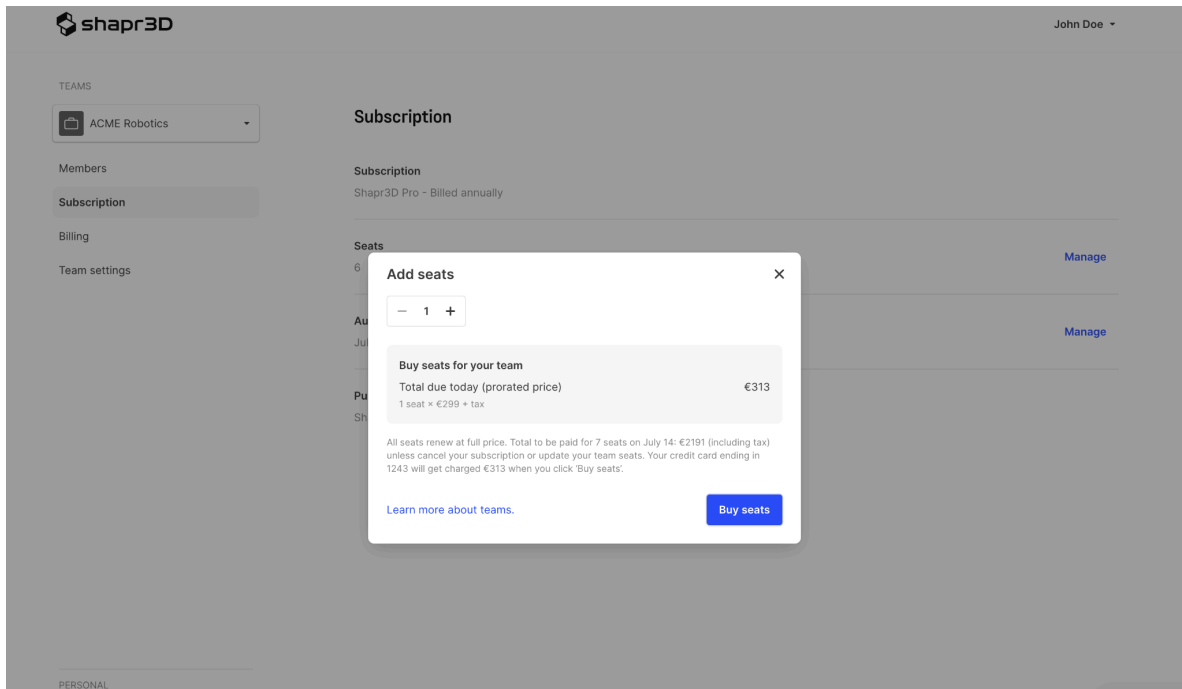
PERSONAL

Account settings

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**.

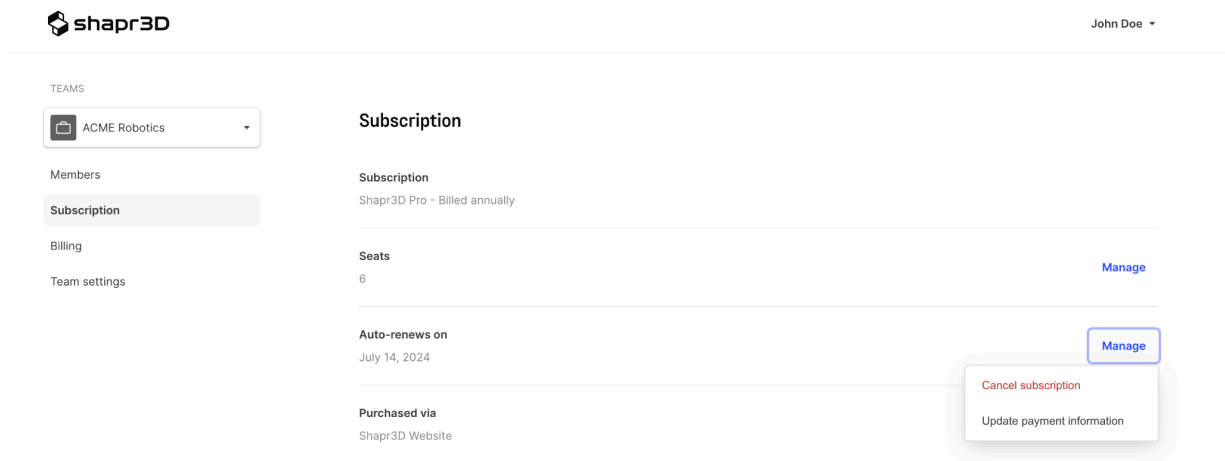


Manage auto-renewal

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.

Data security and privacy

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Shapr3D values transparency and user privacy. Refer to our [Privacy Policy](#) and [Data Processing Agreement page](#) to learn about your rights, what kind of personal data we collect, and other related information about these types of policies.

Read on to learn additional information about other types of user data that Shapr3D collects, Sync encryption, and how to opt out from sending data.

User data collection

For more information, visit our [Privacy Policy](#) and [Data Processing Agreement page](#).

Sync data frequently asked questions (FAQ)

Find answers below for frequently asked questions about [Shapr3D Sync](#).

Can I integrate my own server instead of the cloud?

Not at the moment.

How are my projects encrypted in the cloud?

Currently, there is no end-to-end encryption; projects are encrypted at rest and in transit. The Shapr3D team has access to your projects but the team can only process your content to:

- Grant you access to the services and any corresponding services, such as Sync and conversion services.
- Maintain the services so they are in working order, such as troubleshooting or responding to incidents.
- Comply with any legal requirements or governmental requests.
- Research how the services are used so the team can improve them.

Note: If you explicitly allow the Shapr3D team to access your content, that means the team can access your projects for the purposes shared above.

Is two-factor authentication available for users and required for Shapr3D?

Two-factor authentication is not available for end-user authentication. It is required for Shapr3D employees who have access to the backend, including Shapr3D Sync.

Where are the storage servers physically located? Under what circumstances will you allow third parties to access them?

Storage servers are located in the Ireland region of Amazon Web Services. Shapr3D does not allow third parties to access projects stored via Shapr3D Sync, except for situations in which the company must comply with any legal requirements or governmental requests.

Shapr3D crashed and I lost my work. Can my project still be restored?

Yes. 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).

Opt out from sending data

When you first launch Shapr3D, a popover appears that gives you the option to stop the software from sending your data. If you choose to enable sending usage data, you can opt out at any time in [Settings or Preferences](#).

When you opt out from sending data, our analytics providers will cease collecting all relevant feature usage data.

However, there are exceptions:

- Crash reports will still be sent to [Crashlytics](#) so we can identify and fix crashes quickly. This crash report does not include any information that can be used, in itself, to identify individual users.
- The data required for registering to Shapr3D (your email address, CAD experience, purpose of using CAD, and field of expertise) will still be collected. Otherwise, we can't send you onboarding emails. To stop receiving emails from us, click **Unsubscribe** in any of our emails. If you want to delete your account and user data, [submit a request to our support team](#).

For more information about Shapr3D's policies on data usage, go to [Privacy Policy](#).

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](#)
- [Sharing via Webviewer](#)
- [Sharing via web embed](#)

Shapr3D Sync

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Shapr3D Sync is a Shapr3D Cloud feature that allows you to open, view, and edit the latest version of your projects on any device tied to your account. With Shapr3D Sync, your projects will always be backed up in Shapr3D Cloud, and you'll have access to them from when they were last synced whenever you're 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 work on the same project and switch between multiple devices seamlessly.** With Shapr3D Sync, your projects are always up-to-date on all devices, and any changes you make are automatically synced. You can pick up where you left off on any device.
- **You have a new device and you want to transfer your projects from your old device to your new one.** With Shapr3D Sync, you can simply sign in from any device to access your projects.
- **You want an option to keep a cloud backup for your projects.** With Shapr3D Sync, you don't have to worry about losing your projects if something happens to your device.

Tip

Shapr3D Sync is optional and you can opt out of it anytime.

Enable or disable Sync

If you're a newly signed-up user, **Sync** is enabled by default. You can disable Sync from [Settings or Preferences](#) or from the [Dashboard](#).

Note: Disabling Sync doesn't delete previously sync'd projects, but updates you make no longer sync across all devices.

Important

If you disable Sync, projects you create without Sync will only be [stored locally](#) and cannot be recovered if you uninstall Shapr3D. You also won't be able to access your projects on other devices and use Webviewer to share your projects.

Sharing via Webviewer

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Shapr3D Webviewer is a feature of Shapr3D Cloud that allows you to share your projects via a web link. Share stunning 360° views of your projects with anyone. Send a Webviewer link, and the recipient can view the project using any modern browser.

Read more

To learn more, visit [Webviewer](#).

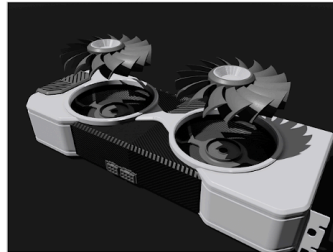
Sharing via web embed

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Website embedding is another feature of Shapr3D Cloud that allows you to embed your projects to your website via HTML code. Simply copy the HTML embed code to your website for all your viewers or clients to see.

[Home / Shop / Electronics Cooling Fans](#)

[< Prev](#) | [Next >](#)



Electronics Cooling Fans

SKU: 36421531913191

\$20.00

Quantity

Add to Cart

PRODUCT INFO

Compatible Graphics Cards:

- o ZX 9010
- o ZX 9020
- o XZ 9030

RETURN & REFUND POLICY

SHIPPING INFO



Specifications:

- o Voltage: DC 12V
- o Fan diameter: 75mm & 85mm
- o Power Connector: 4 Pin

Graphics Card

AR

[Try Shapr3D](#)

View in 3D



Read more

To learn more, visit [Embed your projects.](#)

Local storage

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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

[\[Section link\]](#) | [\[Table of contents\]](#) | [\[Back\]](#) | [\[Next\]](#)

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.** The Webviewer feature is 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 sideways. 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

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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?](#)

Navigation

Dashboard, Items Manager, and managing projects
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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

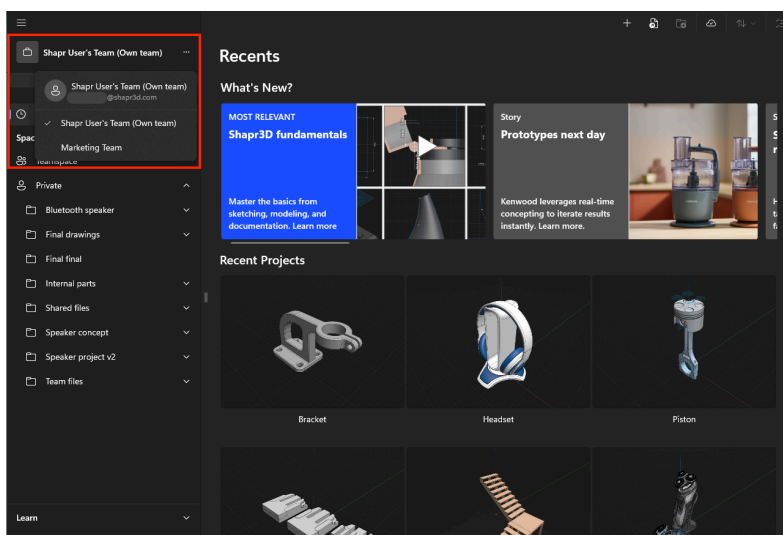
The team name displays the team dashboard you're currently in. If you're a member of multiple teams, you can only view the projects from the team currently selected in the sidebar.

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](#).

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.

Spaces

Under **Spaces**, you can find the library of your shared and private projects under **Teamspace** and **Private**.

Teamspace

The Teamspace contains team projects synced across all team members' accounts.

In Teamspace:

- Everyone can view and edit projects.
- Folders or projects moved into the Teamspace automatically sync across all team member's accounts and devices.
- Only you will retain access to a project moved to your private space.
- You can import your active team projects, team archive, part library, and more to the Teamspace to share with your colleagues.

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.

Private

Private space is where you can create projects only accessible to you.

In private space:

- Only you retain access to a project moved to your private space.
- You can create your own projects and move ongoing projects to and from Teamspace.
- Your projects aren't visible to your colleagues and only sync within your own devices.

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

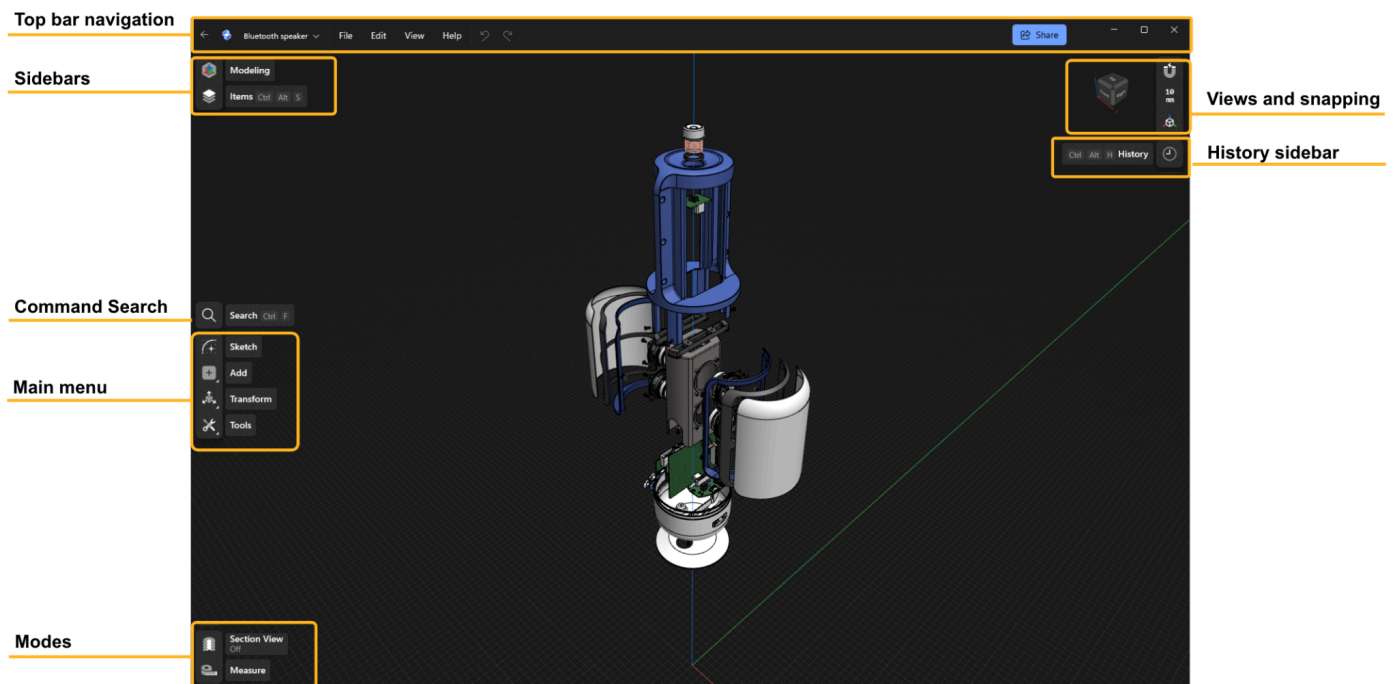
[\[Article link\]](#) | [\[Table of contents\]](#) | [\[Back\]](#) | [\[Next\]](#)

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

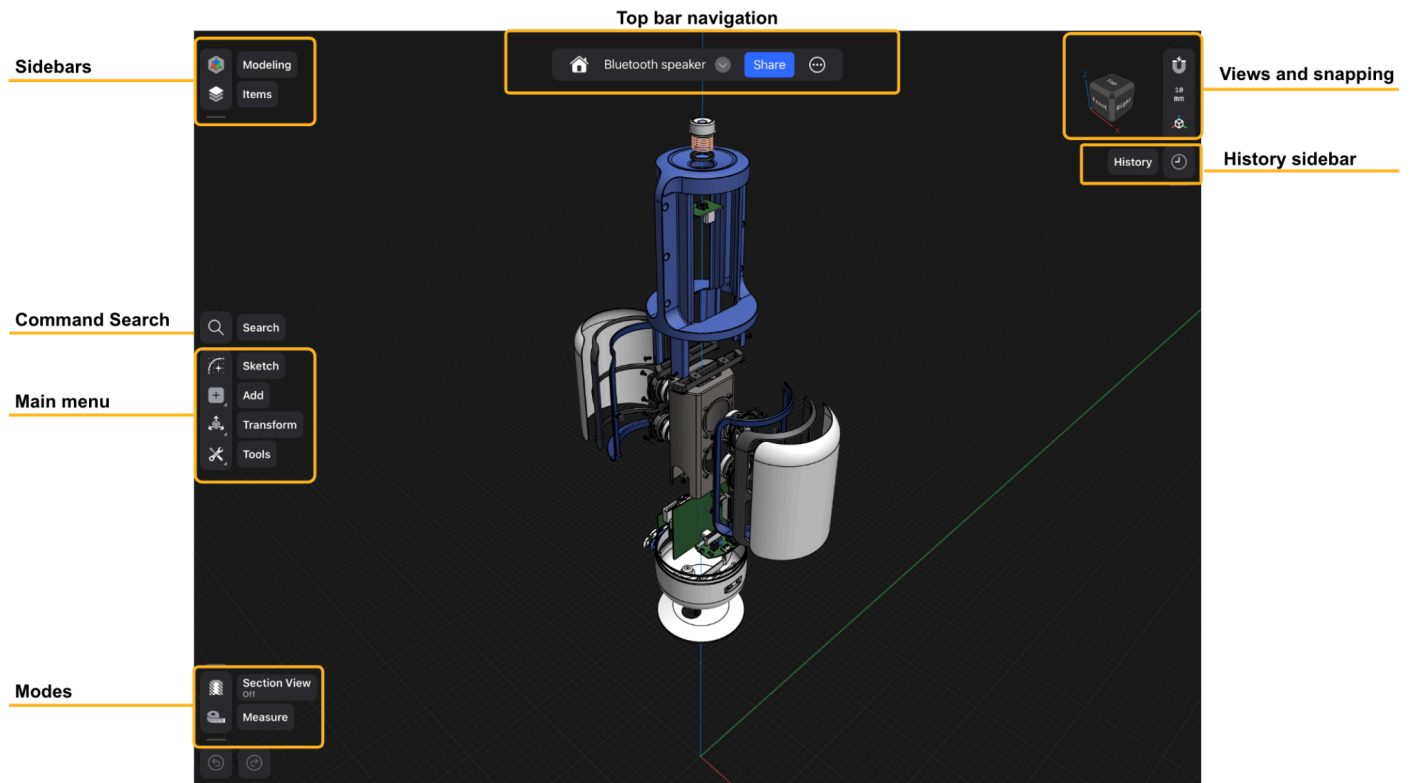
In the modeling space, you can find the following areas:

- [Top bar navigation](#)
- [Sidebars](#)
- [Main menu and Command Search](#)
- [Modes](#)
- [Views and snapping](#)

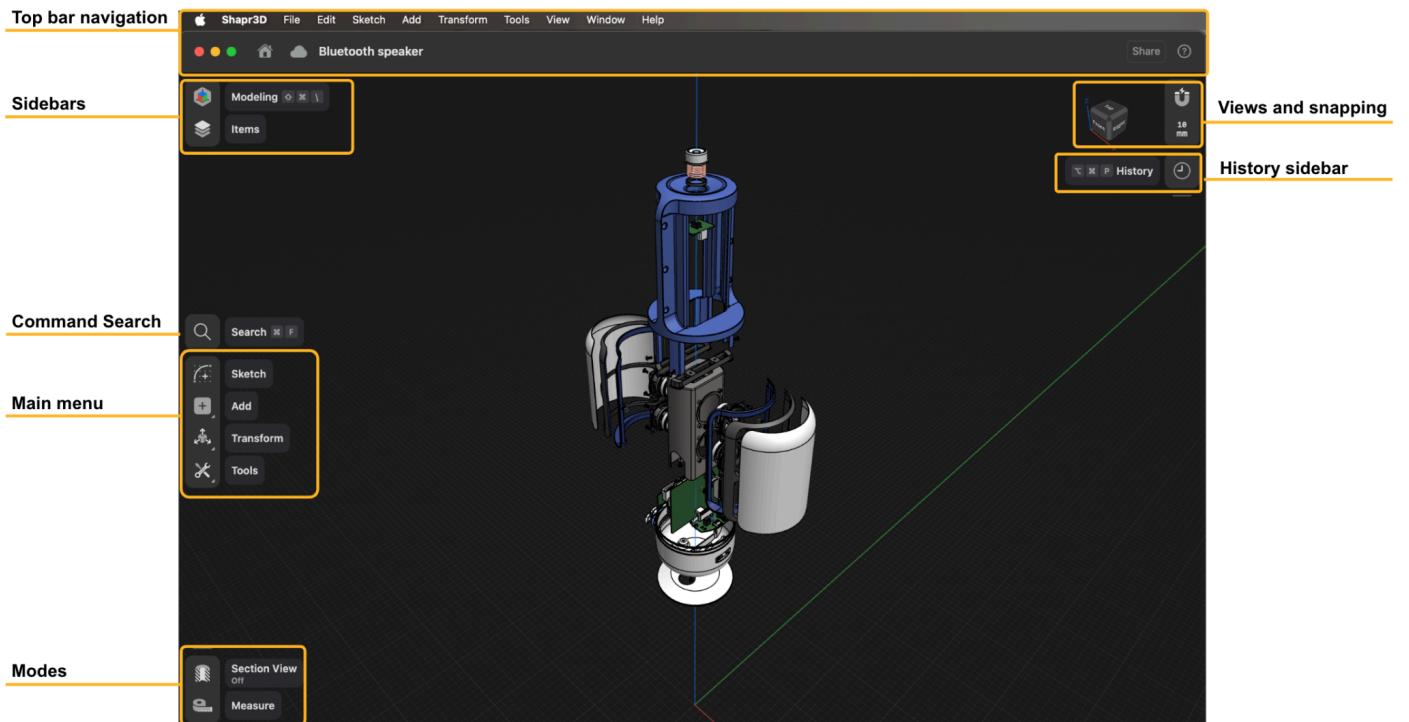
Windows



iPad










Mac







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 create a [Webviewer link](#) that you can share with anyone, and an **Export** tab where you can export your projects.
- **Help**  – Shortcut to the **Help** menu.
- **More Settings**  – 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](#).
- **Add**  – Opens the add menu. To learn more about each tool, visit [Add 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**.

History

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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:

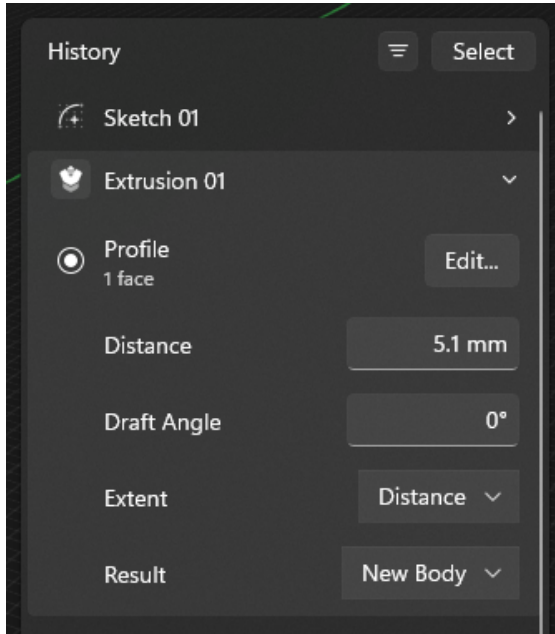
- **Insert Breakpoint** – Adds a breakpoint after the step. Steps after the breakpoint are temporarily disabled and will not be visible on your model. To remove a breakpoint, simply hover over the breakpoint and then select the X.
- **Suppress/Unsuppress** – Suppressing a step hides the step from the 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

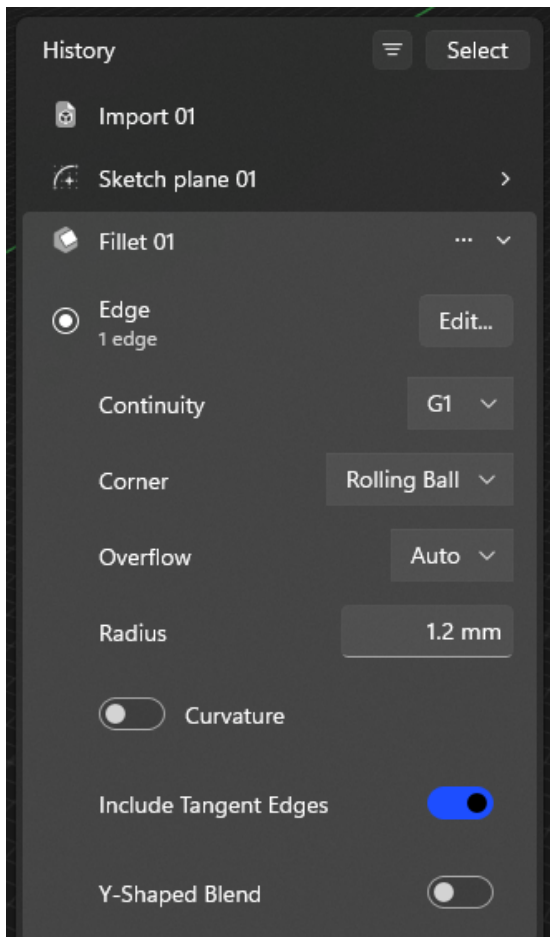
You can view or modify parameters you defined for relevant steps in their equivalent **History** card.

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

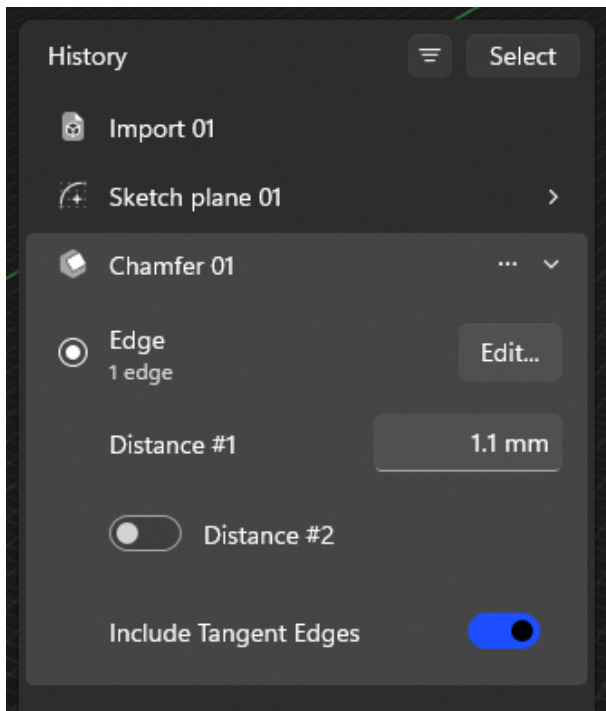
Extrude



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:

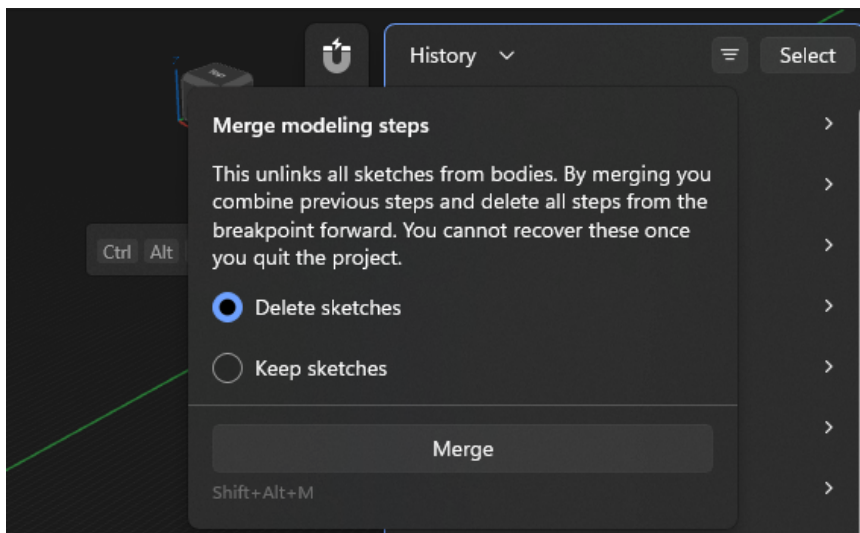
- All your sketches unlink or break their connections from bodies.
- You combine all previous steps and delete all steps before and after the breakpoint.
- You have the option to delete or keep sketches

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.
4. To finish, select **Merge**.



Learn more by watching the video below:

<https://youtu.be/rmmizsmc1Hk>

Manage your projects and folders

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You can manage all your work under **Spaces** in the Shapr3D Dashboard sidebar.

Under **Spaces**, you can find the library of your shared and private projects under **Teamspace** and **Private**.

Create a new project



You can create a new project directly from the **Teamspace** or **Private** space.

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 the **Teamspace** or **Private** space:

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

Export a project to .shapr file

To export a project to a .shapr file:

1. Secondary click (right-click) or long-tap on a project.
2. Select **Save as .shapr file**.


Rename a project

To rename a project:

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

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**.

Create and manage folders

You can create folders under **Teamspace** or **Private** 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 **Teamspace** or **Private** space.
- 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

A deleted project can't be recovered. If Sync is enabled and you delete a synced project, the project will be deleted from all devices. You can keep a copy of your project before deleting it by [exporting it](#).

Enable Sync

If you have Sync disabled, you can enable Sync directly from the Projects page by selecting  in the top-right.

To learn more, visit [Shapr3D Sync](#).

General settings

Settings, peripherals, and accessories

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Navigation Presets

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You can change your Shapr3D navigation shortcuts in **Navigation Presets**. Choose from the different presets available and find the best one according to your preference.

Access Navigation Presets

To access your navigation presets:

1. Go to [Settings or Preferences](#).
 - **iPadOS:** Select **Navigation Preset**
 - **macOS:** Select **Navigation Presets...**
 - **Windows:** Select **Navigation Presets...**
2. Select your preferred preset under **Shapr3D Presets** or choose among the available CAD software presets under **Popular Presets**.
Note: iPadOS only supports Shapr3D Default and Shapr3D Classic.
3. To save the changes, select **Apply** (macOS) or **Apply preset** (Windows).

Shapr3D presets

Below is a list of all presets available in Shapr3D.

Recommended

- Shapr3D Default
- Shapr3D Classic

Popular presets (macOS and Windows only)

- 3ds Max
- AutoCAD
- Blender
- Cinema 4D
- Fusion 360
- Lego Studio
- OnShape
- Rhino
- SketchUp
- SolidWorks

Settings or Preferences

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Settings or **Preferences** is where you can view your Shapr3D software version and set up or update your preferred settings for Shapr3D such as account settings, navigation preferences, and Sync.

Access Settings or Preferences

- **Windows:**
 - From the **Dashboard**, select **Settings**.
 - From the modeling space, go to **File > Settings**.
- **iPadOS:** Tap the **Settings** icon at the bottom of your screen
- **macOS:** To access **Preferences**, go to the Mac menu bar > **Shapr3D > Settings...**

Shapr3D settings

You can access the following settings:

Note: 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.
- **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** – Enables **Command Search** or **Hotkeys**.
- **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](#).

Graphics

- **Rendering Quality (macOS)** – Choose your preferred rendering quality.
- **Anti-Aliasing (Windows)** – Choose your preferred anti-aliasing setting.
- **Resolution (Windows)** – Use the slider to adjust your screen resolution.

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.
- **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 Across Devices** (iPadOS and Windows) – Toggle to enable or disable **Sync**.
- **Status** (macOS) – Select to **Enable Across Devices** or **Disable Across Devices**. When you enable **Sync**, the **Sync Projects** option appears and auto-refreshes to show you the status of when your project was last synced. You can also select this option to manually sync your projects.
- **Sync Projects** – Appears when **Sync** is enabled and auto-refreshes to show you the status of when your project was last synced. You can also 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 iPadOS, 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.

Peripherals and accessories

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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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Access **View/Views** or **Appearance** to update your preferred view and appearance settings for your modeling space such as grid position, projection, and surface analysis.

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:** Go to the Mac menu bar and then select **View**.

Views

- **Default View** – Select to reset your view.
- **Views** – Select between **Top**, **Bottom**, **Front**, **Back**, **Right**, and **Left** as your preferred plane during 2D modeling.
Note: You can also use the [Orientation Cube](#) to choose a plane or reset your view.
- **Grid Position** – Choose between **XY Plane**, **YZ Plane**, and **ZX Plane** as your preferred planar orientation.

Appearance

- **Field of view** – Fine-tune your view or projection by selecting your preference on a scale of 0 (orthographic) to 90 (perspective) degrees.
- **Show Hidden Edges** – Toggle to enable or disable hidden edges. This setting makes the bodies transparent, and highlights and selects their edges. This is useful if you have a complex geometry and you want to see what's "inside."
- **2-Finger Rotation** – Enable this option to use two fingers to rotate your view without moving the position of the camera.
- **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](#).
- **Surface Analysis** – Toggle to **Show Zebra**. When zebra is enabled, you can see how the surfaces meet and determine the surface continuity (G0, G1, and G2). Under **Direction**, you can specify whether the stripes should be horizontal or vertical.

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.

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.

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.

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)	<p>Mouse/Magic Mouse: Secondary click (right-click) and drag the mouse</p> <p>Trackpad:</p> <ul style="list-style-type: none">• iPadOS/macOS: Pan with two fingers on the trackpad.• Windows: Hold down CTRL + Shift and pan one finger on the trackpad. <p>Note: These controls use the Shapr3D Default navigation preset. To learn more, go to Navigation presets.</p>	
Pan (Move camera)	<p>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.</p> <p>Magic Mouse: Hold the Shift key and use secondary click (right-click) to drag the mouse.</p> <p>Trackpad:</p> <ul style="list-style-type: none">• 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. <p>Note: These controls use the Shapr3D Default navigation preset. To learn more, go to Navigation presets.</p>	
Zoom camera	<p>Mouse: Scroll with the mouse wheel</p> <p>Magic Mouse: Hold the CTRL key and pan up/down with your finger on the mouse.</p> <p>Trackpad: Pinch in or out with two fingers on the trackpad</p> <p>Note: These controls use the Shapr3D Default navigation preset. To learn more, go to Navigation presets.</p>	
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

Tool/Action		Gesture/Shortcut/Hotkey	
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
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		

Tool/Action	Gesture/Shortcut/Hotkey
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 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

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 the endpoint and then Delete in the menuTap 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: Use 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

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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

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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

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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

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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](#).

Basic selection

The methods for making selections are listed in the table below:

	Touch and pen	Keyboard and mouse/trackpad
Select a face, sketch, or edge	Tap once	Click once
Select an entire body	Tap twice	Double-click
Select connected sketch elements	Tap twice	Double-click
Select multiple items, one by one	Tap each item without interruption.	Press the Shift key and click each item.
Select all	Long-tap the screen and drag across the space to activate the Area Selection tool.	macOS: ⌘A Windows: Ctrl+A
Deselect all	Tap an empty area.	Click an empty area. Windows/macOS: Esc iPadOS: ⌘ .
Select an area	Long-tap the screen and drag across the space to activate the Area Selection tool.	Click in the modeling space and drag the pointer across the space to activate the Area Selection tool.

Tip

You can also make selections from the [Items Manager](#).

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.

Notable points

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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

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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.

Aside from switching between spaces, the Project Sidebar also provides you a shortcut to add, rename, or delete 2D Drawings.

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

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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 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: 

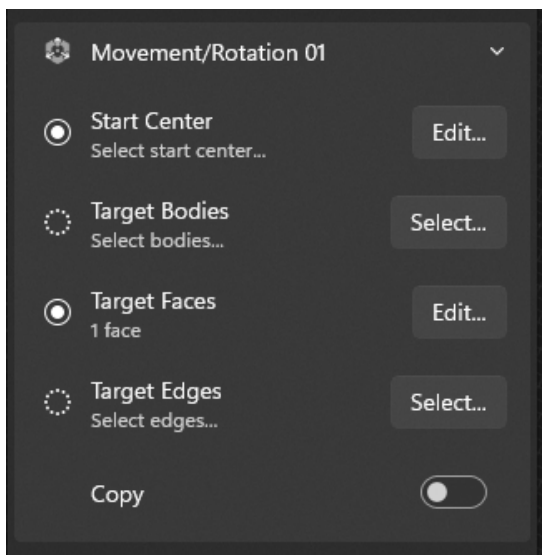
To use the **Copy** badge:

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 your 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.

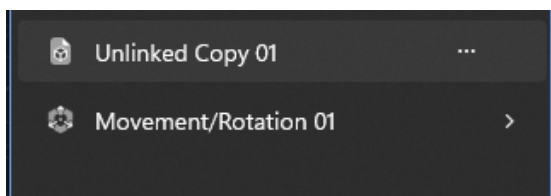
History settings

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

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When you model, you can quickly define or calculate numerical values such as length, radius, or distance, using a keyboard or the built-in numpad.

Keyboard input

You can directly define values in both 2D and 3D views for dimension labels or annotation badges.

To define and calculate numerical values while you're modeling:

1. Click the dimension label.
Note: Depending on your settings or current state, you may need to click an item or [gizmo arrow](#) to show the dimension label.
2. Use your keyboard to input the new value.
3. To make calculations, input the values and basic mathematical operators.
4. **Optional:** For multiple dimension labels available in the current position, you can move between them by pressing the **Tab** key.
5. To finalize the value, press **Enter**.

Using the numpad

The numpad, or numpad calculator, can also be used to define numerical values for your model. You can enter values directly in the numpad or calculate a value by using the calculator functions.

Tip

Some dimension labels, such as the quantity for patterns, display a simplified numpad that does not include the calculator functions.

Activate the numpad

To activate the numpad while you're modeling:

- **Keyboard and mouse/trackpad:** Click the dimension value to select it and then click it again to activate the numpad.
Note: Depending on your settings or current state, you may need to click an item or [gizmo arrow](#) to activate the numpad.
- **Touch and pen:** Tap the dimension value to activate the numpad.

Input and calculate values in the numpad

- **Keyboard and mouse/trackpad:** Click or type values and mathematical operators and then press **Enter** or click the check mark to finalize the value.
- **Touch and pen:** Tap the buttons to input or calculate a value and then tap the check mark to finalize the value.

Read more

To learn more about dimension labels, visit [Editing sketch dimensions](#).

Boolean operations

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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 points, patterns, and dimensioning

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States of sketch points

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You can determine information about points in a sketch by understanding how different states are represented in the modeling space. The different states of sketch points are:

- **Unconnected**  – Unconnected points are free to move in any way within the sketch.
- **Connected**  – A connected point can indicate the following states:
 - The point is connected to another sketch point. Both points will move together when modified.
 - The point is connected to a line or curve with a [Coincident constraint](#). The point can move along the line or curve or along a projection of the line or curve.
 - The point is connected to the center of a line with a [Midpoint constraint](#). The point will remain at the line's center if the line's length or position is modified.
- **Locked**  – A locked point is fixed and cannot be moved.

Tip

You can modify the states of sketch points by adding and removing constraints. For more information, visit [Adding and removing constraints](#).

Sketch pattern constraint

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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.

Learn more by watching the video below:

 [Shapr3D Manual - Using pattern constraints | Sketching](#)

Read more

To learn more about the Sketch Pattern tool, visit [Pattern \(Sketch\)](#).

Editing sketch dimensions

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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.

Learn more by watching the video below:

 [Editing sketch dimensions | Sketching](#)

Using sketch planes

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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.

Learn more by watching the video below:

 [Shapr3D Manual - Defining sketch planes | Sketching](#)

Changing a sketch plane

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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

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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

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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.

Customize your material

You can customize a material color and adjust other settings 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.

- **Scale** – Drag the slider to change the scale 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.
- **Hexadecimal** – If you know the hex value of a color you want to apply, enter the value and press the **Return** or **Enter** key to apply that color.
- **Color Swatches** – You can store colors that you frequently use. Choose a color in the **Slider** or sample a color in your modeling space with the **Pick Face Color** button, and then select an empty slot in the **Color Swatches**. The chosen or sampled color will be saved. To remove a color, secondary click (right-click) it and select **Delete**.

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

In the example below, we're change the orientation of the Y-axis to better match the orientation of the material to the overall body of the model.

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.

Customize an environment

To customize an environment:

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.
 - **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](#).






2D Drawings

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The **2D Drawings** space is where you can create 2D Drawings or technical drawings for your 3D models.

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  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**

Learn more by watching the video below:

[▶ Shapr3D Manual - Create new drawings | 2D Drawings](#)

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.

Learn more by watching the video below:

[▶ 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  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.











Learn more by watching the video below:

 [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**.

Learn more by watching the video below:

 [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 Drawings** 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**.

Learn more by watching the video below:

[▶ 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**.

Learn more by watching the video below:

[▶ 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**.

Learn more by watching the video below:

 [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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Drawing lines and arcs with a pen | Sketching](#)

Read more

Learn how to create a [line](#) or [arc](#).

Line

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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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Drawing lines | Sketching](#)

Read more

For information on how to edit the dimensions of a sketch, visit [Editing sketch dimensions](#).

Arc

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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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Drawing arcs | Sketching](#)

Read more

For information on how to edit the dimensions of a sketch, visit [Editing sketch dimensions](#).

Spline

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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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Drawing splines | Sketching](#)

Read more

To learn more about constraints, visit [Constraints overview](#). For information on how to edit the dimensions of a sketch, visit [Editing sketch dimensions](#).

Rectangle

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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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Drawing rectangles | Sketching](#)

Read more

For information on how to edit the dimensions of a sketch, visit [Editing sketch dimensions](#).

Circle

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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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Drawing circles](#)

Read more

For information on how to edit the dimensions of a sketch, visit [Editing sketch dimensions](#).

Ellipse

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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.

Note: To learn more about sketch projections, visit [Project](#).

Learn more by watching the video below:

 [Shapr3D Manual - Drawing ellipses | Sketching](#)

Read more

For information on how to edit the dimensions of a sketch, visit [Editing sketch dimensions](#).

Polygon

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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.
Note: To learn more about sketch projections, visit [Project](#).

Learn more by watching the video below:

 [Shapr3D Manual - Drawing polygons | Sketching](#)

Read more

For information on how to edit the dimensions of a sketch, visit [Editing sketch dimensions](#).

Offset Edge (Sketch)

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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.

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:
 - **Loop** – Offsets a single edge from a sketch element.
 - **Single** – Offsets a loop 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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Offsetting sketches | Sketching](#)

Read more

For information on using Offset Edge with sketch profiles and edges of 3D bodies, visit [Offset Edge \(3D\)](#).

Move/Rotate (Sketch)

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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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Moving and rotating sketches | Sketching](#)

Tip

To find more video tutorials, visit our [Shapr3D Manual YouTube Playlist](#).

Pattern (Sketch)

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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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Creating sketch patterns | Sketching](#)

Text

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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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Adding text | Sketching](#)

Project - Sketches

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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**.

Learn more by watching the video below:

 [Shapr3D Manual - Trimming sketches | Sketching](#)

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**.

Learn more by watching the video below:

 [Shapr3D Manual - Deleting sketches | Sketching](#)

Constraints menu

Constraint overview, settings, and types

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Constraints overview

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Constraints can be added to sketch elements to create geometric controls and relationships. By defining constraints, you can add information to your sketches that control their behavior when they are modified. For example, you can add a [Horizontal/Vertical](#) constraint to a line to ensure it remains aligned with the axes of the sketch plane.

When working in a sketch, valid constraints for selected elements are highlighted in the menu at the right. You can also choose to have constraints created automatically for you while sketching by adjusting options in [Constraint Settings](#).

Learn more by watching the video below:

 [Shapr3D Manual - Introduction to constraints | Sketching](#)

Constraints Settings

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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.

Learn more by watching the video below:

 [Shapr3D Manual - Introduction to constraints | Sketching](#)

Adding and removing constraints

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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. [Select](#) the sketch element(s) you want to constrain.
2. Select the constraint type..

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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Adding and removing constraints | Sketching](#)

Parallel


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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 lines in the sketch
2. From the constraints menu, select **Parallel**.

Learn more by watching the video below:

 [Shapr3D Manual - Types of constraints | Sketching](#)

Perpendicular

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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.
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.

Learn more by watching the video below:

 [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. Select two sketch elements, either a line and a curve, or two curves.
2. 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.

Learn more by watching the video below:

 [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. [Select](#) an endpoint.
2. Select another element.
3. 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.

Learn more by watching the video below:


 [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


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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. [Select](#) an endpoint and line.
2. Select **Midpoint** from the constraints menu or drag an endpoint onto the center of a line.

Learn more by watching the video below:


 [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. [Select](#) arc or circle elements.
2. From the constraints menu, select **Concentric**.

Learn more by watching the video below:

 [Shapr3D Manual - Types of constraints | Sketching](#)

Horizontal/Vertical

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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 line or lines.
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](#).

Learn more by watching the video below:

 [Shapr3D Manual - Types of constraints | Sketching](#)

Equal


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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, such as multiple lines or multiple arcs.
2. Select **Equal** from the constraints menu.

Learn more by watching the video below:

 [Shapr3D Manual - Types of constraints | Sketching](#)

Symmetry


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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. 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.

Learn more by watching the video below:


 [Shapr3D Manual - Types of constraints | Sketching](#)

Disconnect


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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.
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.

Learn more by watching the video below:


 [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.

Learn more by watching the video below:

 [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

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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.

Learn more by watching the video below:

 [Shapr3D Manual - Making construction geometries | Sketching](#)

Add menu

Tools under the Add menu

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Understanding construction geometry

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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 Axis

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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 **Add > Construction Axis**.
For other methods of accessing this tool, visit [Accessing tools](#).
2. Select **Type** then choose a construction axis type.
3. Select appropriate references for the type of axis you chose (for details, see [Construction Axis types](#)).
4. 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. Select the type of construction axis you want to define from the **Type** menu. The tooltip at the top of the modeling space will provide you with directions on what you need to do to create the type of axis you have chosen.

Between 2 Points

Use the **Between 2 points** type of construction axis 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.

2-Plane Intersection

Use the **2-Plane Intersection** type of construction axis 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.

Cylindrical/Conical

Use the **Cylindrical/Conical** type of construction axis 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.

Along Edge

Use the **Along edge** type of construction axis 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.

Perpendicular to Face

Use the **Perpendicular to Face** type of construction axis to create an axis that is perpendicular to a selected planar element and through a selected point. Follow these steps to create this type of axis:

1. Select a face, plane, or sketch profile.
2. Select **Next**.
3. Select an existing point or select anywhere in the modeling space to define the point the axis will run through.
4. **Optional**: Drag the points to reposition the axis. The points snap to the grid, sketch geometry, or body geometry.
5. 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.

Construction Plane

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Use the **Construction Plane** tool 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 **Add > Construction Plane**.
For other methods of accessing this tool, visit [Accessing tools](#).
2. Select **Type** and choose a construction plane type.
3. Select appropriate references for the type of plane you chose (for details, see Construction Plane types).
4. Select **Done**.

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

You can define a **Construction Plane** in several different ways. Select the type of construction plane you want to define from the **Type** menu. The tooltip at the top of the modeling space will provide you with directions on what you need to do to create the type of plane you have chosen.

Offset

Use the **Offset** 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. Select **Next**.
3. Drag the arrows to define the offset distance 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 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** construction plane type 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** – 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.

3 Points

Use the **Through 3 Points** construction plane type 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.

On Curve at Point

Use the **On Curve at Point** construction plane type 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 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.

Perpendicular to Edge

Use the **Perpendicular to Edge** construction plane type 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.
2. Select **Next**.
3. Select an existing point or select anywhere in the modeling space to define a point where the plane will be located.
4. **Optional**: Drag the automatically generated point to reposition the plane. The point will snap to the grid, sketch geometry, or body geometry.
5. 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.

Along Edge at Angle

Use the **Along Edge at Angle** construction plane type 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.
2. Select **Next**.
3. Select a face, plane, or sketch profile.
4. Select **Next**.
5. Drag the arrows to define the angle or use the dimension label to enter a value.
6. 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.

Parallel to Face

Use the **Parallel to Face** construction plane type to create a plane that is parallel 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.
2. Select **Next**.
3. Select an existing point or select anywhere in the modeling space to define a point where the plane will be located.
4. **Optional**: Drag the automatically generated point to reposition the plane. The point will snap to the grid, sketch geometry, or body geometry.
5. 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.

Image

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The **Image** tool under the **Add** menu is another way to access **Import**. To learn more about importing images, go to [Import](#).

File

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The **File** tool under the **Add** menu is another way to access **Import**. To learn more about importing files, go to [Import](#).

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: 

Learn more by watching the video below:

 [Shapr3D Manual - Copying items | Transform](#)



Move/Rotate (3D)

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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  .
Note: The **Copy** badge appears as blue when selected .
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.

Learn more by watching the video below:




 [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**  .
Note: The **Copy** badge  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.

Learn more by watching the video below:

 [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.

Learn more by watching the video below:

 [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.


Learn more by watching the video below:

 [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. Use the following steps 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.

Learn more by watching the video below:

 [Shapr3D Manual - Rotate Around Axis | Transform](#)

Align

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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 [Accessing 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** – Click/tap **Edit...** or **Select...** to choose a different target and then select **Done** to finish.

Learn more by watching the video below:

 [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 [Accessing 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.

Learn more by watching the video tutorial below:

 [Shapr3D Manual - Mirror | Transform](#)

Tools menu

Modeling tools

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Offset Face

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You can use the **Offset Face** tool to offset faces or bodies. When you offset a face, the tangent faces are automatically included in the offset operation. You can use this tool to add or remove thickness, or modify the size of features such as holes.

Using 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.
4. Select **Done**.

Note: When using **Offset Face** from the [adaptive menu](#), select an empty area of the grid to complete the tool.

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:

- **Face** – Click/tap **Edit...** or **Select...** to choose a different face to offset and then select **Done** to finish.
- **Distance** – Enter your preferred face offset distance.

Learn more by watching the video below:

 [Shapr3D Manual - Offset Face | Tools](#)

Offset Face vs. Extrude

While the tools look similar, **Offset Face** cannot be used to create a body from a closed sketch, while [Extrude](#) creates 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.

Chamfer/Fillet

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You can use the **Chamfer/Fillet** tool to add an angled face or radius at the edges of your model.

The type of geometry that is created 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.

Creating a chamfer

To create a basic 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:
 - **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 Chamfer** – A chamfer that is set back at different distances from the selected edge.
3. [Select](#) the edge or edges you want to chamfer.
4. [Use the gizmo](#) to chamfer the selected edges.
Note: Negative values will define a chamfer, positive values will define a fillet radius.
5. Select **Done**.
Note: When using **Chamfer/Fillet** from the [adaptive menu](#), select an empty area of the grid to complete the tool.

Creating a fillet

A fillet produces a radius at a selected edge. 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 condition is referred to as a **Rolling Ball**. These settings can be adjusted to fine-tune a fillet (see [Understanding fillet settings](#)).

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 edge or edges you want to fillet.
3. [Use the gizmo](#) to fillet the selected face.

Note: Negative values will define a chamfer, positive values will define a fillet radius.

4. Select **Done**.

Note: When using **Chamfer/Fillet** from the [adaptive menu](#), select an empty area of the grid to complete the tool.

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

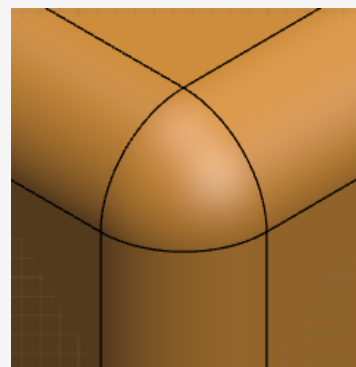
This dimension label defines the radius for a default circular fillet. If you make changes to the profile or continuity type, this value represents the setback distance from the original edge, rather than a true radius value.

Settings badge

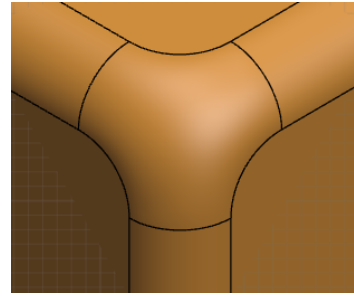
Select the Settings badge to access the menu with the **Corners** and **Continuity** options.

Corners

Rolling Ball This type of corner has a constant radius. This is the default corner type.

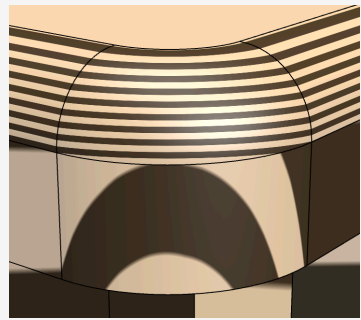


Setback This type of corner sets back the edges where the fillets begin blending into the corner. The blended face has a variable radius.

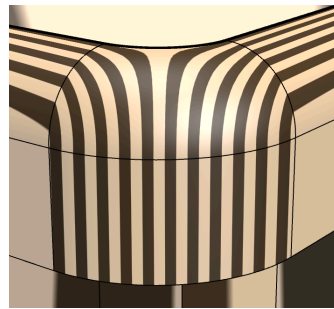


Continuity

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.
- **Continuity** – Select the dropdown to choose between a G1 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.
- **Radius** – Enter your preferred radius for the fillet.
- **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.

Learn more by watching the video below:

 [Shapr3D Manual - Chamfer/Fillet | Tools](#)

Extrude

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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 **Extrude** tool can result in geometry that is:

- A new body.
- Added to an existing body.
- Removed from an existing body, creating a hole.
- The intersection of the extruded profile with an existing body.

The type of extrude created is automated based on geometric conditions, but it can be modified if necessary.

Using 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. If necessary, adjust the **Extrude** type using the boolean badge next to the dimension label.
Note: For more information, see [Understanding default Extrude types](#).
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.
- **Distance** – Enter your preferred distance for the extrusion.
- **Draft Angle** – Enter your preferred draft angle for the extrusion.
- **Extend** – Select the dropdown to choose between **Distance**, **To Object**, or **To Point** as the extent type for the extrusion and select **Select...** beside **Object** to select the object where you want to extend the extrusion.
- **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.

Learn more by watching the video below:

 [Shapr3D Manual - Extrude | Tools](#)

Understanding default Extrude types

The default type of extrude that is created is based on geometric conditions. Here is a summary of the types of extrudes and when they are triggered:

	Description:	The default type when:
New Body 	Creates a new standalone body.	A sketch profile is extruded that is not in contact with any existing body.
Union 	The extruded geometry will be added to an existing body. The edges where the extruded faces and the existing body meet will become shared edges.	<ul style="list-style-type: none">• A face is selected for the extrude and pulled away from the existing body.• A sketch profile that is attached to a face is selected and pulled away from the existing body.
Subtract 	The extruded geometry will be removed from an existing body, producing a hole or void.	<ul style="list-style-type: none">• A face is selected for the extrude and pushed into the existing body.• A sketch profile is selected and pushed into an existing body.
Intersect 	The result of the extrude will be the volume where the profile overlaps existing body geometry.	This type of extrude must be selected from the 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.

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.

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.

Learn more by watching the video below:

 [Shapr3D Manual - Shell | Tools](#)

Loft

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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.

To learn more, explore the sections below.

- [Understanding Loft elements](#)
- [Using the Loft 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.

Using 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 points.
5. If necessary, select the loft guide curves.
6. **Optional:** For lofts with periodicity, you can blend or smoothen the first and last profile by turning on **Periodic Loft** in the [History sidebar](#).
7. Select **Done**.
Note: When using **Loft** from the [adaptive menu](#), select an empty area of the grid to complete the tool.

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.

Learn more by watching the video below:

 [Shapr3D Manual - Loft | Tools](#)

Union

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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.

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.

Learn more by watching the video below:

 [Shapr3D Manual - Union | Tools](#)

Subtract

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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.

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 at the left. 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.
5. Select **Done**.

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.

Learn more by watching the video below:

 [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.

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. If necessary, adjust the **Keep Originals** setting at the left with any of the following:
 - **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.
5. Select **Done**.

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.

Learn more by watching the video below:

 [Shapr3D Manual - Intersect | Tools](#)

Split Body

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You can use the **Split Body** tool to cut apart 3D bodies with construction planes, grid planes, sketch profiles, faces, or images. This allows you to split a body into multiple bodies without removing any material.

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. The splitting tool will be highlighted in purple and marked with a target badge.
Splitting tools are projected through the body being split. Therefore, the splitting tool does not need to be in direct contact with the body.
Note: You can select the badges to redefine selections and toggle them on and off.
4. If necessary, adjust the **Keep Originals** setting at the left with any of the following:
 - **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.
5. Select **Done**.

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:

- **Target** – Click/tap **Edit...** or **Select...** to choose a different body you want to split and then select **Done** to finish.
- **Surface** – Click/tap **Edit...** or **Select...** to choose a different surface 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.

Learn more by watching the video below:

 [Shapr3D Manual - Split Body | Tools](#)

Revolve

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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.

Using the Revolve tool

To create a **Revolve**, you will need a profile and an axis of revolution. The profile can be a closed sketch or a face. The axis can be an axis of the grid, a construction axis, a sketch line, or a linear edge. The **Revolve** tool rotates the profile about the axis to generate the solid geometry. By default, the profile is revolved by 360°, but you can modify the angle value as necessary. To generate a helical shape, you can specify a height for the revolve action.

To use the **Revolve** tool:

1. Ensure you have the elements necessary for the **Revolve**, including:
 - Profile: A sketch profile or face representing half of the finished body's cross-section.
 - Axis: An axis, line, or linear edge element at the center of rotation for the body.
2. Go to **Tools > Revolve**.
For other methods of accessing this tool, visit [Accessing tools](#).
3. Select the profile.
4. Select the axis.
5. **Optional:** [Use the gizmo](#) to adjust the angle for the revolve.
6. If necessary, adjust the height for the revolve to create a helical shape (see [Creating helical bodies](#)).
7. Select **Done**.

Note: When using **Revolve** from the [adaptive menu](#), select an empty area of the grid to complete the tool.

Creating helical bodies

Helical bodies are created by defining a height value within the **Revolve** tool.

To create a helical body:

1. Ensure you have the elements necessary for the **Revolve**, including:
 - Profile: A sketch profile or face representing half of the finished body's cross-section.
 - Axis: An axis, line, or linear edge element at the center of rotation for the body.
2. Go to **Tools > Revolve**.
For other methods of accessing this tool, visit [Accessing tools](#).
3. Select the profile.
4. Select the axis.
5. [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.
6. [Use the gizmo](#) to adjust the height for the revolve action.
7. Select **Done**.


Note: When using **Revolve** from the [adaptive menu](#), select an empty area of the grid to complete the tool.

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.

Learn more by watching the video below:

 [Shapr3D Manual - Revolve | Tools](#)

Sweep

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The **Sweep** tool can extrude a profile along a selected path. The profile can be a closed sketch or face. The path, or spine, can be sketch elements or edges of a body. This tool is useful for creating complex bodies such as pipe systems, cables, wires, and other shapes with a uniform cross-section. For shapes that have varying cross-sections, use the [Loft](#) tool.

To use the **Sweep** tool:

1. Ensure you have the elements necessary for the **Sweep**, including:
 - **Profile**: A sketch profile or face that you will extrude.
 - **Spine**: Sketch elements or edges that the profile will follow.
2. Go to **Tools > Sweep** tool.
For other methods of accessing this tool, visit [Accessing tools](#).
3. Select the profile or profiles.
4. Select **Next**.
5. Select the spine or spines.
6. Select **Done**.

Tip

When using Sweep from the [adaptive menu](#), select the profile and all spines before activating the tool.

History settings

To manage history settings for the Sweep action, find the step for your specific sweep and then view or modify the following:

- **Profile** – Click/tap **Edit...** or **Select...** to choose a different sketch profile for the Sweep action and then select **Done** to finish.
- **Path** – Click/tap **Edit...** or **Select...** to choose a different path for the Sweep action and then select **Done** to finish.

Learn more by watching the video below:

 [Shapr3D Manual - Sweep | Tools](#)

Replace Face

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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.

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. If necessary, adjust the **Other Side** setting at the left.
5. 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.
- **Other Side** – Turn on to extend the face replacement to the other side of the replacing face.

Learn more by watching the video below:

 [Shapr3D Manual - Replace Face | Tools](#)

Offset Edge (3D)

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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.

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:
 - **Loop** – Offsets a loop of edges from a 3D body.
 - **Single** – Offsets a single edge 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**.

Learn more by watching the video below:

 [Shapr3D Manual - Offset Edge \(3D\) | Tools](#)

Read more

For information on using Offset Edge within a sketch, visit [Offset Edge \(Sketch\)](#).

Project

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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.

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.

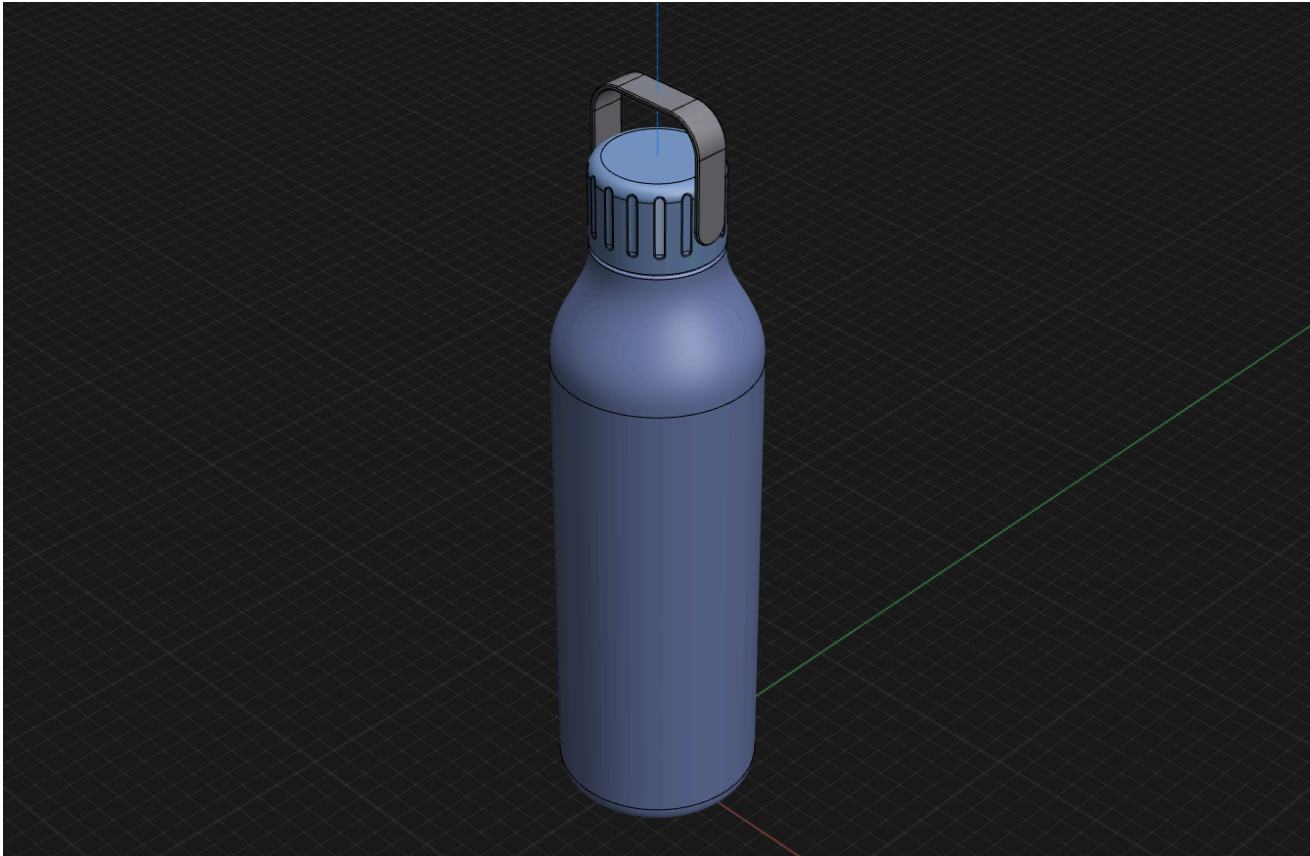
When you project your items as edges to a surface of a body, the resulting edge projection:

- 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

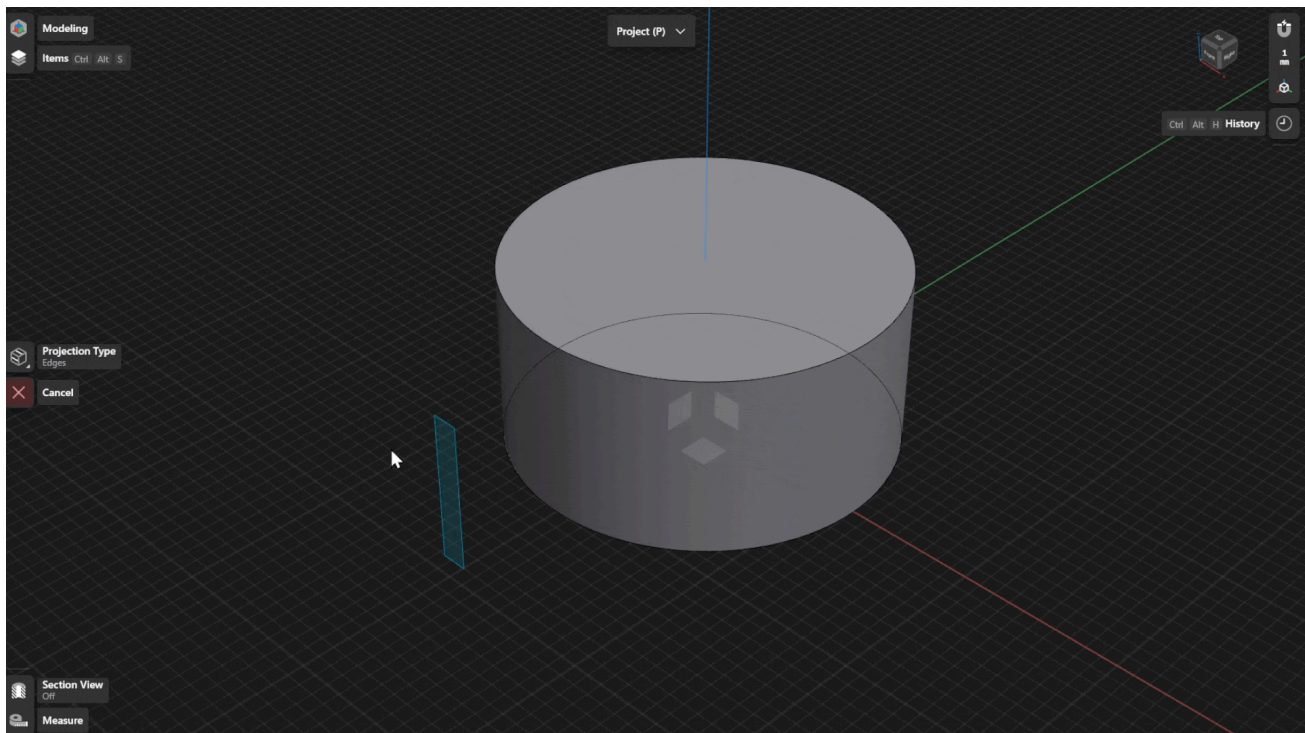
Check out some examples below to learn how you can use edge projections in your modeling workflow.

Example 1

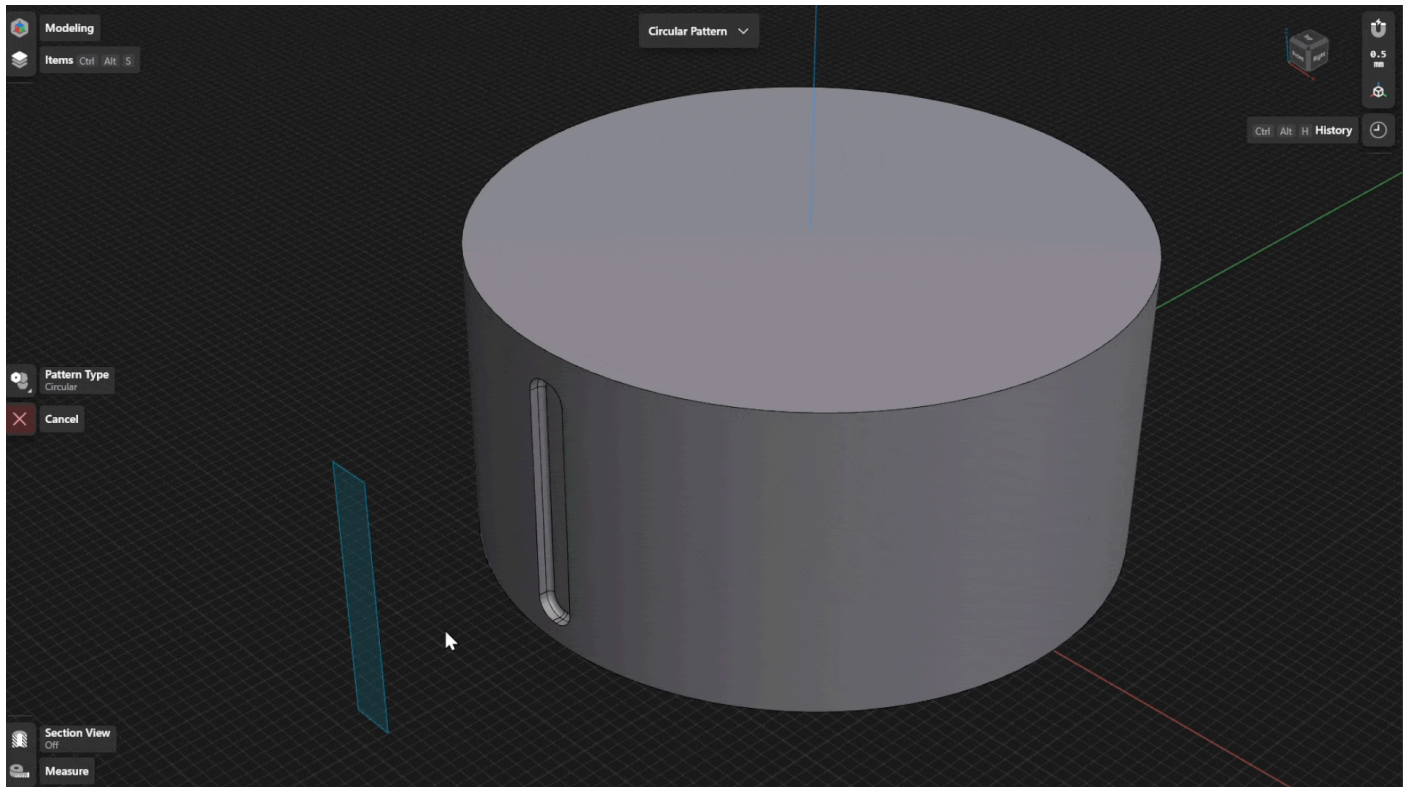
In the example model, we have a bottle with groove designs on the cap.



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:

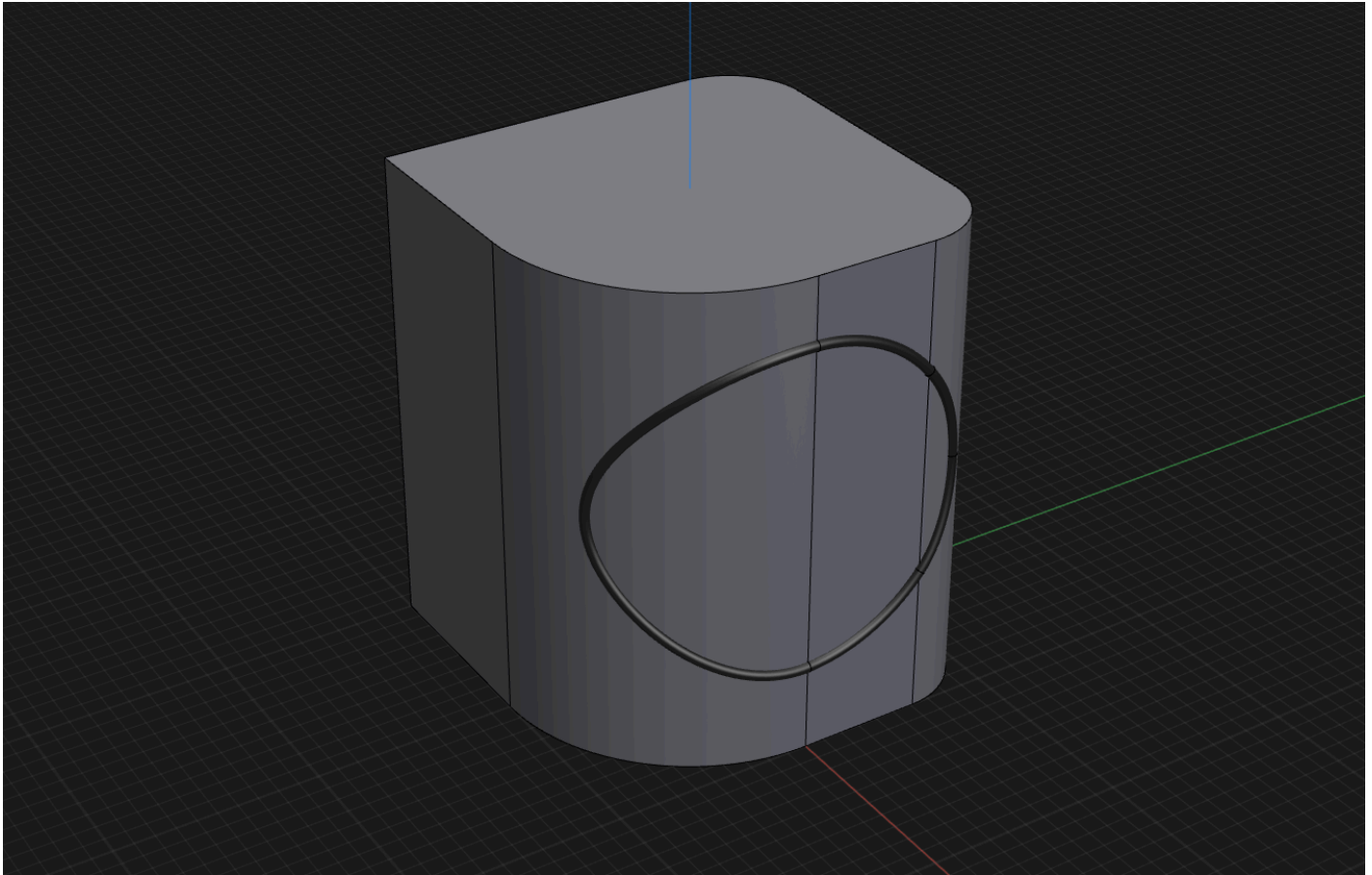


Lastly, you can use the [Pattern](#) tool to create a circular pattern around the cylinder.

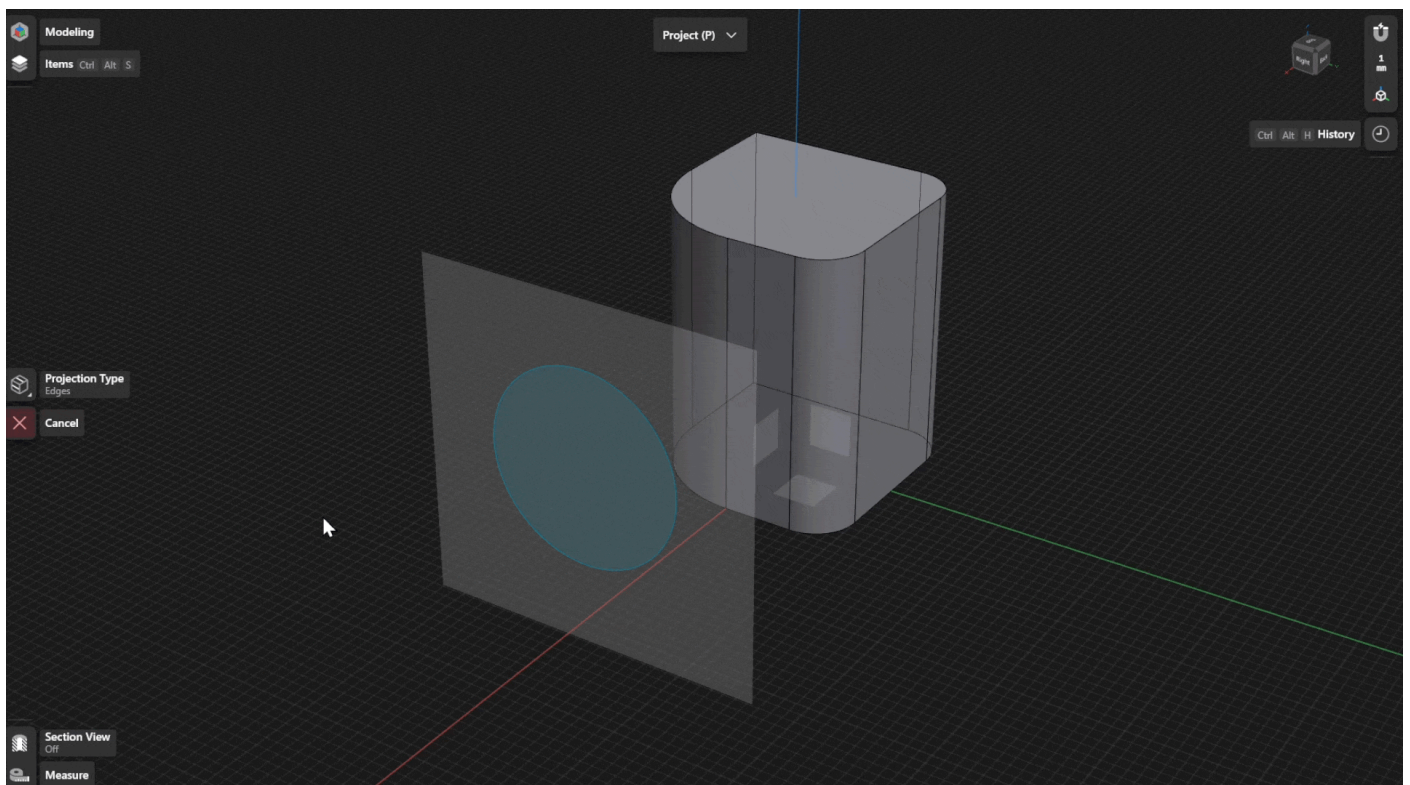


Example 2

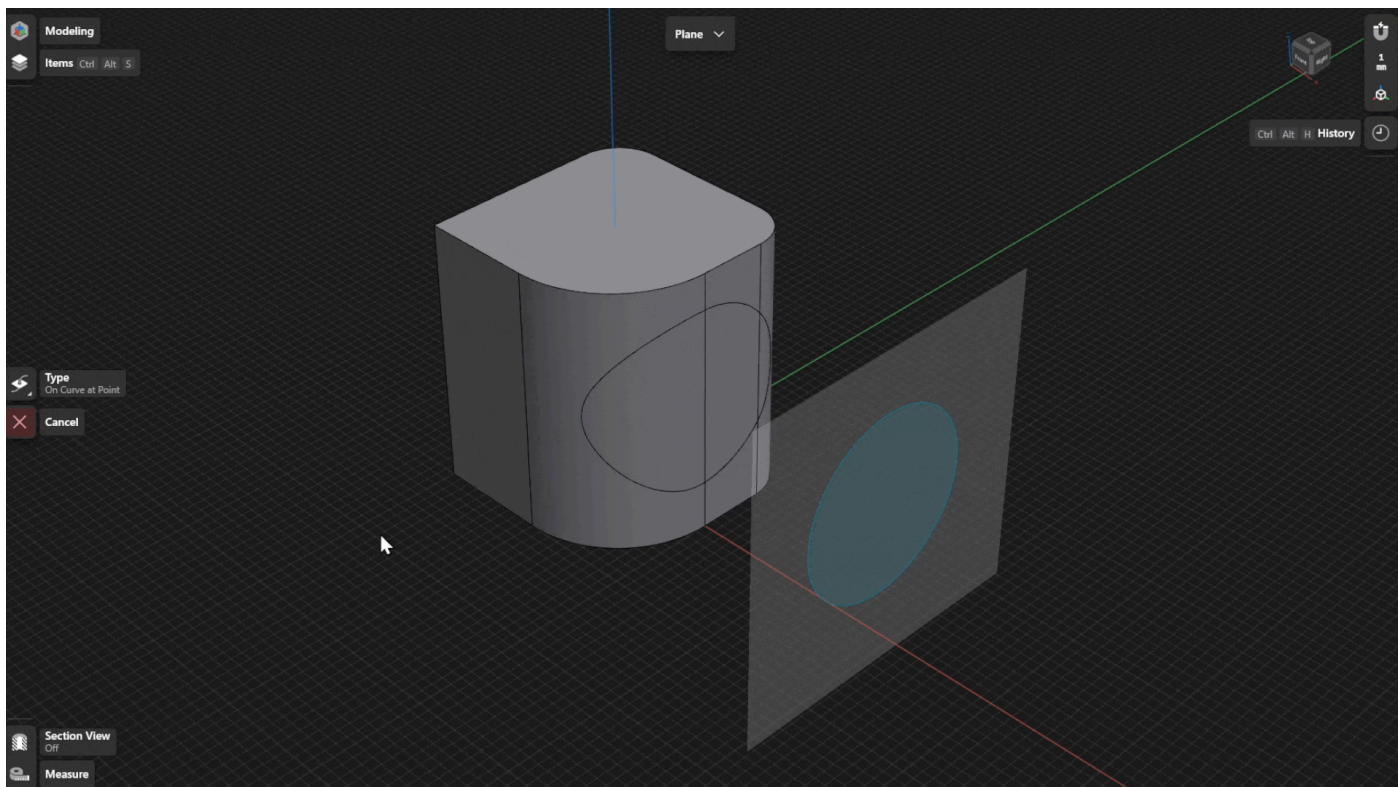
In the example model below, we have a circular ridge on a non-planar surface of the body.



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.

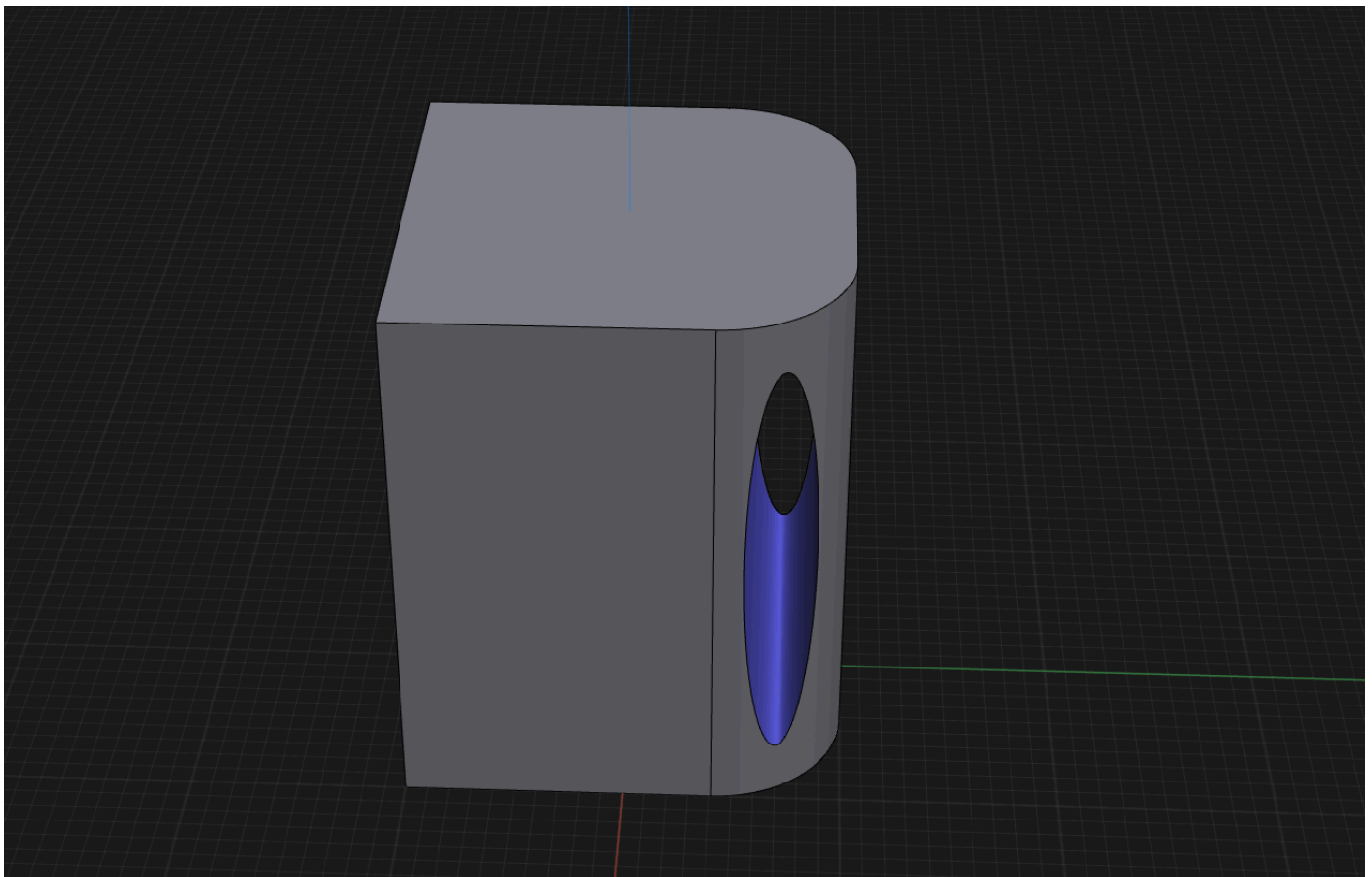
When you project your items as sketches to a surface of a body, the resulting sketch projection:

- is treated like a regular sketch item that can be hidden in the [Items Manager](#)
- can be used as reference sketches similar to how you treat sketches when directly sketching on a face

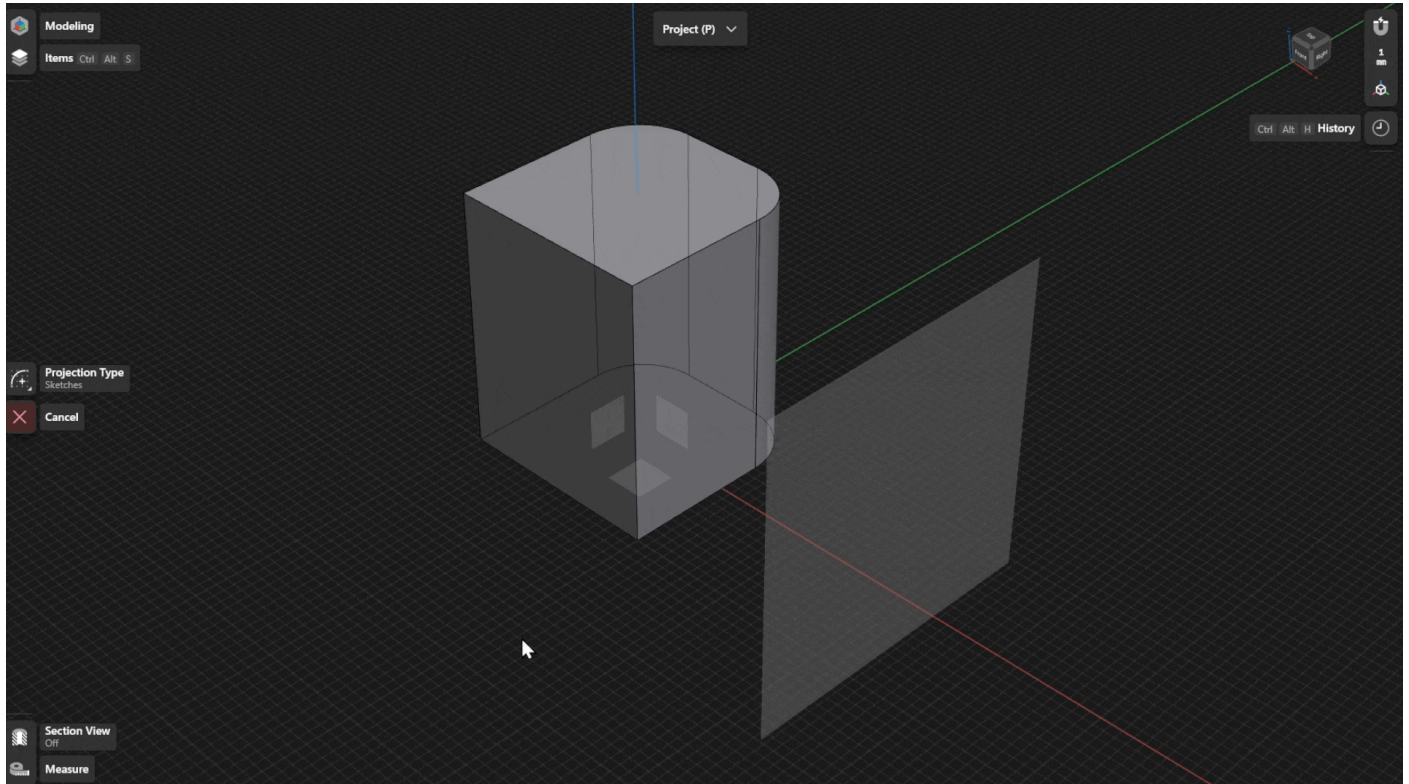
Check out the example below to learn how you can use sketch projections in your modeling workflow.

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.

Learn more by watching the video below:

 [Shapr3D Manual - Project | Tools](#)

Visualize

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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

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A section view shows the internal parts of 3D bodies. It looks as if you cut through an entire length of the model. You can use this view to look at and modify the internal parts of your model.

You can create a section view in just a few steps:

1. [Choose a plane](#). This is the section plane that you will use to cut through your 3D model. For example, select **Top** if you want to see the internal section of your 3D model from the top.
2. From the modes in the main menu, select  to turn on **Section view**.
3. Select  to turn off **Section view**.

You can also select a plane or face and then select  to see the section view along the plane or face.

Measure

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Every sketch, face, and body in Shapr3D is built with precise parameters. **Measure** allows you to view the exact measurements of elements in the models you create. While in Measure mode, a panel pops up where you can select and pin the exact measurements of selected elements. The pop-up panel stays open while you work on your model, and you can drag this panel anywhere on your screen.

With **Measure**, you can:

- [View measurements](#)
- [Pin measurements](#)
- [Add point-to-point measurements](#)
- [Highlight measurements](#)









View measurements

1. From the modes in the main menu, select **Measure** to open the pop-up panel.
2. Select the items in your model that you want to measure.
3. View the measurement of the selected items under **Current Measurements** in the Measurements panel.

Tip

For quick reference, select items to see 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** ^{min}  – 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** $\overset{\text{cntr}}{\Delta}$ – 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** $\overset{\text{prl}}{\Delta}$ – 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.

Pin measurements

A pinned measurement adds an annotation to your model to help you visualize or communicate the measurement better.

Note: You can only see the annotation for your pinned measurements while in Measure mode.

To pin a measurement:

1. Under **Current Measurements**, select  beside the measurement.
2. View your pinned measurements anytime under **Pinned Measurements**.
3. Optional: To delete or unpin a pinned measurement, go back to the pop-up panel and select  beside the pinned measurement.

Add point-to-point measurements

The point-to-point measurement feature lets you measure the distance between notable points.

To add a point-to-point measurement:

1. From the available modes, select **Measure** to open the pop-up panel.
2. Select **Add Point-to-Point measurement**.
3. Select **Type** to choose a **Measurement type**.
 - **Point-to-Point Distance** - Select 2 notable points to measure their distance.
 - **3-Point Angle** - Select 3 notable points to measure their subtended angle.
4. Optional: Select a face or hover over a face to find notable points.
5. Select the [notable points](#) you want to measure.
6. Optional: You can move already placed points to new notable points.
7. To finish, select **Done**.

Tip


Point-to-point measurements are automatically pinned.

Highlight measurements

You can highlight a current or pinned measurement for better visibility.

To do this, simply hover over a measurement in the measurement panel with your pointer, or tap it with a pen.

Learn more by watching the video below:

 [Shapr3D Manual - Measure your models | Modes](#)

Share your projects

Webviewer, embeds, AR, and more

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Options to share your projects

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Shapr3D has a handful of features that you can use to present and share your projects. Choose among Visualization, Webviewer, embedding, and AR. Learn about the different options below.

Shapr3D Webviewer

Share stunning 360° views of your projects with anyone. Just send a Webviewer link, and the recipient can view the project using any modern browser.

To learn more, visit [Webviewer](#).

Embed to a website

Directly embed your Webviewer link on your website and easily share it with customers or clients.

To learn more, visit [Embed your projects](#).

Augmented Reality (AR)

Shared projects are viewable in Augmented Reality on most iOS or Android phones or tablets.

To learn more, visit [Augmented Reality](#).

Capture tool via Visualization

Speak the same language with your clients: with Visualization, you'll get your concepts through in a blink. Use the capture tool to get snapshots of your model without the grid and other unwanted project elements.

To learn more, visit [Capture your projects via Visualization](#).

Screenshot tool

Customize certain aspects of your model before taking a snapshot.

To learn more, visit [Screenshot tool](#).

Webviewer

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The Webviewer feature gives you the option to share your 3D models and 2D Drawings with anyone via a web link. Your recipient can view your projects directly from any browser on any modern device. You can also password-protect your links to add an extra layer of security.

Create a Webviewer link

To use Webviewer, follow these steps:

1. [Enable Sync](#).
2. Select **Share**.
3. Under the **Share & Embed** tab, select **Create Link**.
If Sync isn't enabled before creating a link, you will see a prompt that asks you to enable sync. Select **Enable Sync** to continue.
4. Once the link is published, you can copy it from the **Share & Embed** tab and share it via any online messaging platform with anyone you want to view your project.
5. To include 2D drawings in your Webviewer link, select **Sharing Options** and then enable **Display 2D drawings**.
6. To change the access permission for your link, select **Link Access**, then choose your preferred options:
 - **Anyone with the link** – Anyone can view your model from the link.
 - **Anyone with the password** – A password is needed to view the link. To set a password, enter a password in the **Add password** field > **Set Password**.
7. If you make any changes to your project, go to **Share** and then select **Publish Changes** to apply your changes. Anyone with the link will see the changes once they refresh the link.
8. To stop sharing your project, go back to **Share**, then select **Stop Sharing**.

View projects via the Webviewer link

3D Models

You can move, rotate, and zoom in and out the models while viewing them through Webviewer. You can also use the Augmented Reality functionality by clicking the **AR** button. To learn more, visit [Augmented Reality](#).

2D Drawings

When you view 2D drawings via a Webviewer link, you can move and zoom in and out the drawing. You also have the option to copy or save the drawing as an image by secondary clicking (right-clicking) and selecting **Save Image As..** or **Copy Image**, or download a PDF copy through the **Download PDF** button.

Embed your projects

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You can embed your Webviewer link on your website to share your projects, portfolio, or team collaboration tool.

To generate an embed code, follow these steps:

1. [Enable Sync](#).
2. Select **Share**.
3. Under the **Share & Embed** tab, select **Create Link**.
Note: If you disabled Sync before creating a link, a prompt pops up that asks you to enable Sync. Select **Enable Sync** to continue.
4. Once the link is published, copy the **HTML embed code** that generates under the **Share & Embed** tab to add to your website.
5. If you make any changes to your project, go to **Share** and then select **Publish Changes** to apply your changes. Anyone with the link will see the changes once they refresh the link.
6. To stop sharing your project, go back to **Share**, then select **Stop Sharing**.

Augmented Reality

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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. Export your project to the Webviewer 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 > Augmented Reality > Preview 3D Model**
 - Select **Share > Share & Embed > Export Project > Preview 3D Model**

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 the Webviewer

This is the easiest way if you are designing on a laptop or a desktop computer without a rear-facing camera.

1. Enable Sync.
2. In your modeling space, select **Share**.
3. Under the **Share & Embed** tab, select **Create Link**.
4. 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.
5. 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 your models via Visualization

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Show, don't tell. With Shapr3D Visualization, make your models realistic and aesthetic with materials and color. With over 100 materials and finish types to choose from, you can customize your models to be exactly how you imagine the end product.

Visualization has a Capture tool that you can use to quickly take pictures or snapshots of your models. Regular screenshots in the modeling space include the grid edges and UI elements while Visualization provides for a higher, more professional quality to capture project previews and deliver realistic snapshots. Easily access these images during presentations or client meetings to speed up the decision-making process.

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**.

Read more

To learn more, visit [Visualization](#).





To learn about other ways to capture images of your models, visit [Screenshot tool](#).

Screenshot tool

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The Screenshot tool for Shapr3D iPad and Shapr3D Mac allows you to customize certain aspects of your model image before taking a picture or screenshot in the modeling space. Easily access these images during presentations or client meetings to speed up the decision-making process.

To use the **Screenshot** tool:

1. Open your project in Shapr3D.
2. Go to **Export**.
3. Under the **Usage** tab, select **Image** to open the **Screenshot** tool.
4. Select **Next**.
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.
6. Click **Capture**  to capture your model.
7. In the **Save** dialog, enter a filename and choose where you want to save your captured image.
8. Select **Save**.

Import and export

Supported file types, exporting, and importing

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Supported file types

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Importing

You can [import files](#) into Shapr3D in these formats:

- **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)

Important

Make sure that your source file is not damaged or corrupted and that your device has enough memory to process the import. The source model should fit the available modeling space in Shapr3D, which is 1 km³. Also, your file should not include any sketches with polylines. When you import files from other software, some lines and curves might not be accurately displayed in Shapr3D.

Exporting

You can [export](#) the following file formats which are compatible with most major CAD software, and the AR ecosystem of Apple:

- **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)
- X_T or X_B
- STEP
- 3MF
- IGES
- OBJ (with colors)
- STL
- GLB
- USDZ

If you have [Sync](#) disabled, you can use our native SHAPR format to transfer models between different Shapr3D platforms.

Tip

If you use Shapr3D Basic, you can only export in low-resolution STL and 3MF. If you're subscribed to the Pro plan, you can export models in all supported formats, and export 2D Drawings in PDF, DXF, or DWG format.


Import

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You can import 2D and 3D projects that you want to view or modify in Shapr3D. To check compatible formats, refer to [Supported file types](#).


Windows

To import a file, follow these steps:

1. Access the **Import** tool through any of the following:
 - From the modeling space,
 - go to **File** and then choose between **Import as a New Project...** or **Import to Current Project...**
 - or, go to the **Add** menu > **Image** or **File**.
 - From the **Dashboard**, go to **Projects** > **Import** .
2. Select the file you want to import > **Open**.
Note: For images, you can [use the gizmo](#) to position the image, and the **Opacity** slider to adjust the transparency. Select **Done** to finish.
3. **Optional:** Use the [Move/Rotate](#) or [Translate](#) tool to reposition your newly imported file.

iPadOS

To import a file, follow these steps:

1. Access the **Import** tool through any of the following:
 - From the modeling space,
 - select **Settings**  > **Import**.
 - or, go to the **Add** menu > **Image** or **File**.
 - From the **Dashboard**, go to **Projects** > **Import**.
2. Select the method for importing the file from the **Import** dialog:
 - **Files** – Select a file from your device or iCloud.
 - **Photos** – Select a photo from your photo gallery.
 - **Camera** – Take a new photo with your iPad camera, then choose **Use Photo** to continue.
Note: For images, you can [use the gizmo](#) to position the image, and the **Opacity** slider to adjust the transparency. Select **Done** to finish.
3. **Optional:** Use the [Move/Rotate](#) or [Translate](#) tool to reposition your newly imported file.

macOS

To import a file, follow these steps:

1. Access the **Import** tool through any of the following:
 - From the Mac menu, Select **File > Import...**
 - From the modeling space, go to the **Add** menu > **Image** or **File...**
 - From the **Dashboard**, go to **Projects > Import** .
2. Select the file you want to import > **Open**.
Note: For images, you can [use the gizmo](#) to position the image, and the **Opacity** slider to adjust the transparency. Select **Done** to finish.
3. **Optional:** Use the [Move/Rotate](#) or [Translate](#) tool to reposition your newly imported file.

Tip

- You can also drag and drop a file or image into the Shapr3D icon in [the dock](#) on your Mac desktop.
- 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**.

History settings

To manage the Import action from the History sidebar:

1. Open **History**.
2. Look for the import step you want to edit.
3. 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

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You can export your Shapr3D projects to use in other CAD software or to share with others.

Export your projects

1. From the modeling space, access the **Export** tool through any of the following:
 - From the file menu, go to **File > Export...**
 - Select **Share > Export tab > Export Project**.
2. Choose between any of the available tabs to export your preferred format.
 - **All formats** – All recommended and other available formats
 - **3D model** – Formats compatible with 3D models: **Shapr3D File, Parasolid model part file, STEP file, IGES file, 3MF file, STL file, OBJ file**.
 - **3D print** – Files compatible with 3D printing: **STL file, 3MF file**
 - **Sketch** – Available sketch files: **DWG file, DXF file, SVG file**
 - **Drawing** – Available drawing files: **DWG file, DXF file, PDF file**
 - **Image** – Available image files: **PNG file, JPEG file**
3. If applicable, customize your [project details](#).
4. Select **Export**.
5. In the **Save As** dialog, enter a file name and choose a location where you want to save your exported file.
6. Select **Save**.

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.

Learn more by watching the video below:

 [Shapr3D Manual - Export your projects](#)

Assemblies

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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

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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

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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

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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: Delete or hide your reference sketch

Once your parts are organized according to the reference sketch, you can delete or hide the reference sketch.

To learn how to delete a sketch, visit [Delete \(Sketch\)](#).

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:

Windows:

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](#).

iPadOS/macOS:

1. From the iPad top bar, go to **Settings** > Export or from the Mac menu bar, go to **File** > **Export**.
2. Go to the **Usage** tab.
3. Select **Sketches**.
4. Select your preferred sketch format.
5. 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

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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

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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

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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 occurred.
- **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 (preferably 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 occurred.

- **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 (preferably 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

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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 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 EDU:** 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 account

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You won't be able to fully access Shapr3D's Customer Support portal if your Shapr3D account's email address is unverified.

Verify your email address and complete your account registration through your account page:

1. Go to the [Shapr3D website](#).
2. Log in to your account.
3. Select your email address at the top right corner.
4. In the drop-down menu, look for the alert about your unverified email address.
5. Select **Resend verification email** in the drop-down menu to receive an email from Shapr3D to complete your account registration.
6. Open the email and select **Verify email address**.

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

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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.

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](#)

Share large cloud files with our support team

If your screen recording results in a larger file or fails to upload in the request form, you can upload your files to [Google Drive](#) or [WeTransfer](#) and include the shareable link in the description of your request.

- Google Drive:
 - [Upload your file](#)
 - [Get a link to share with anyone](#) to ensure that our support representatives can access the files.
- WeTransfer:
 - [Upload your file](#)
 - [Get a link to share with anyone](#) to ensure that our support representatives can access the files.

Find hardware and software specifications

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Knowing the hardware and software specifications of your device is important when you're experiencing any technical issues and need help from our support team. Follow this guide to learn how to find your hardware and software specifications.

Windows

1. Select **Start > Settings > System > About**.
2. Under **Device Specifications** and **Windows specifications**, you can find specifications such as **Processor**, **Installed RAM**, **System type**, and **OS build**.

iPadOS

Go to **Settings > General > About** to find information such as **iPadOS version**, **Model Name**, and **Model Number**.

For more detailed information, visit [Get information about your iPad](#).

macOS

Go to the **Apple menu > About This Mac** to find information such as **Processor** and **Graphics**.

For more detailed information, visit [Get system information about your Mac](#).

EDU license email whitelist guide

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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 EDU 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 EDU license

Follow these steps to get an EDU 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 EDU license through the [Request EDU 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

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Installation issues

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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

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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

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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

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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

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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:

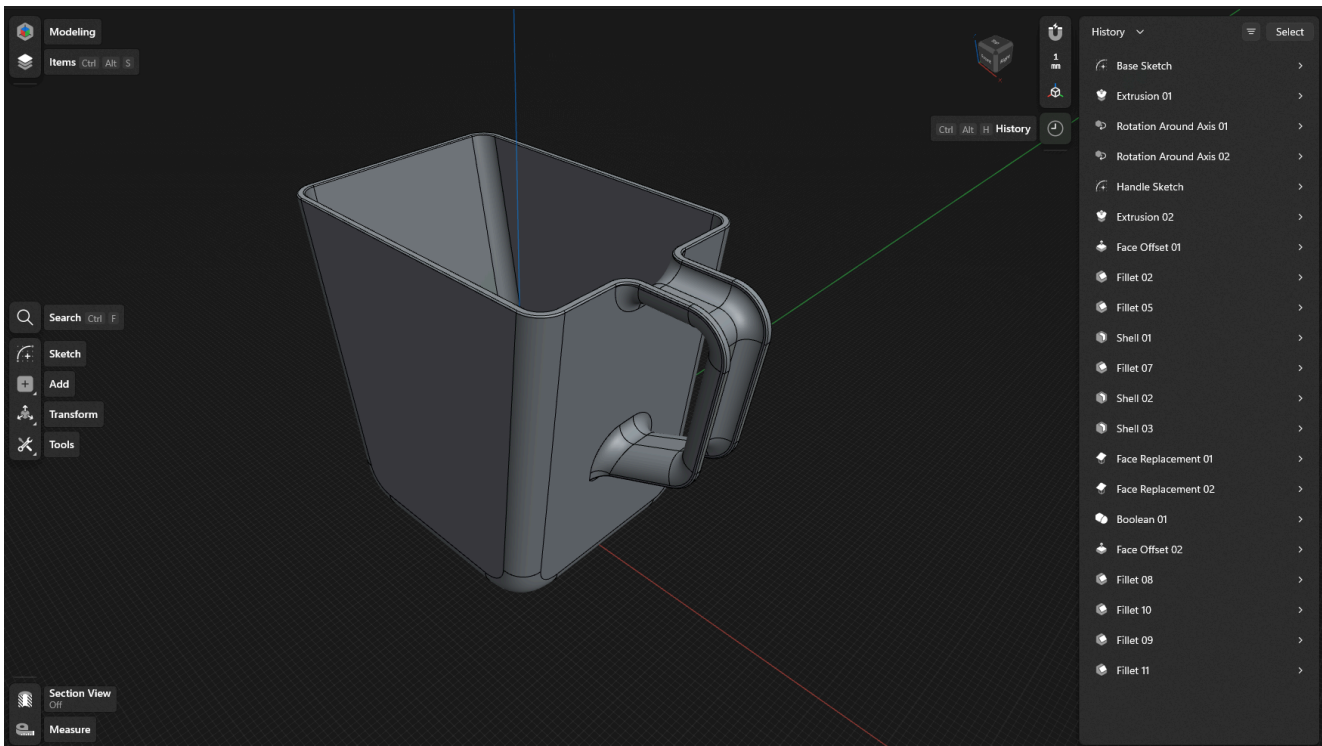
1. From the **History** sidebar, locate the first step with the first error message and then open the expandable card.
2. Select **Fix...** to see what reference is missing. Missing references are shown as an orange hatch in the modeling space.
3. Proceed with any of the following steps:
 - Recreate 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.

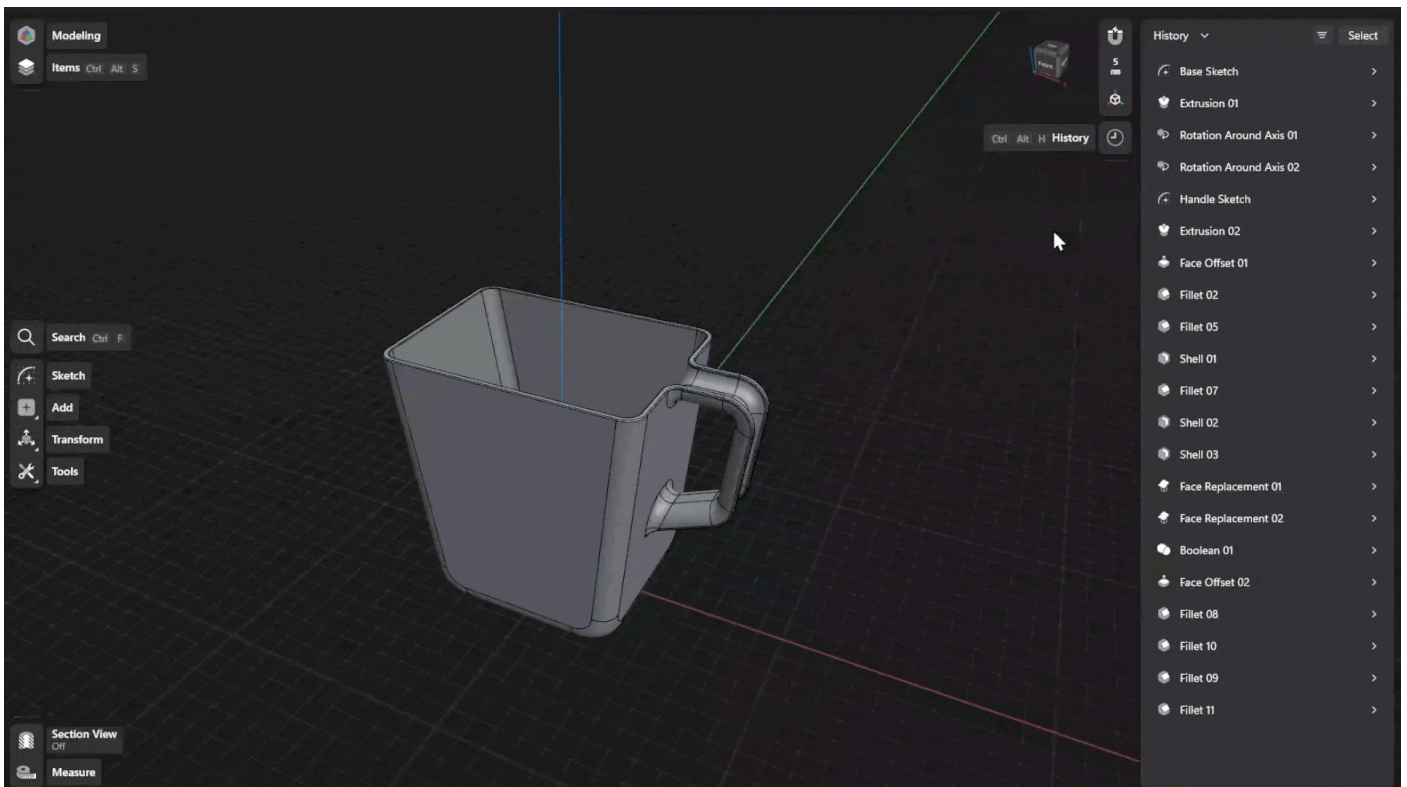
Example #1: Missing model due to a deleted reference step

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:

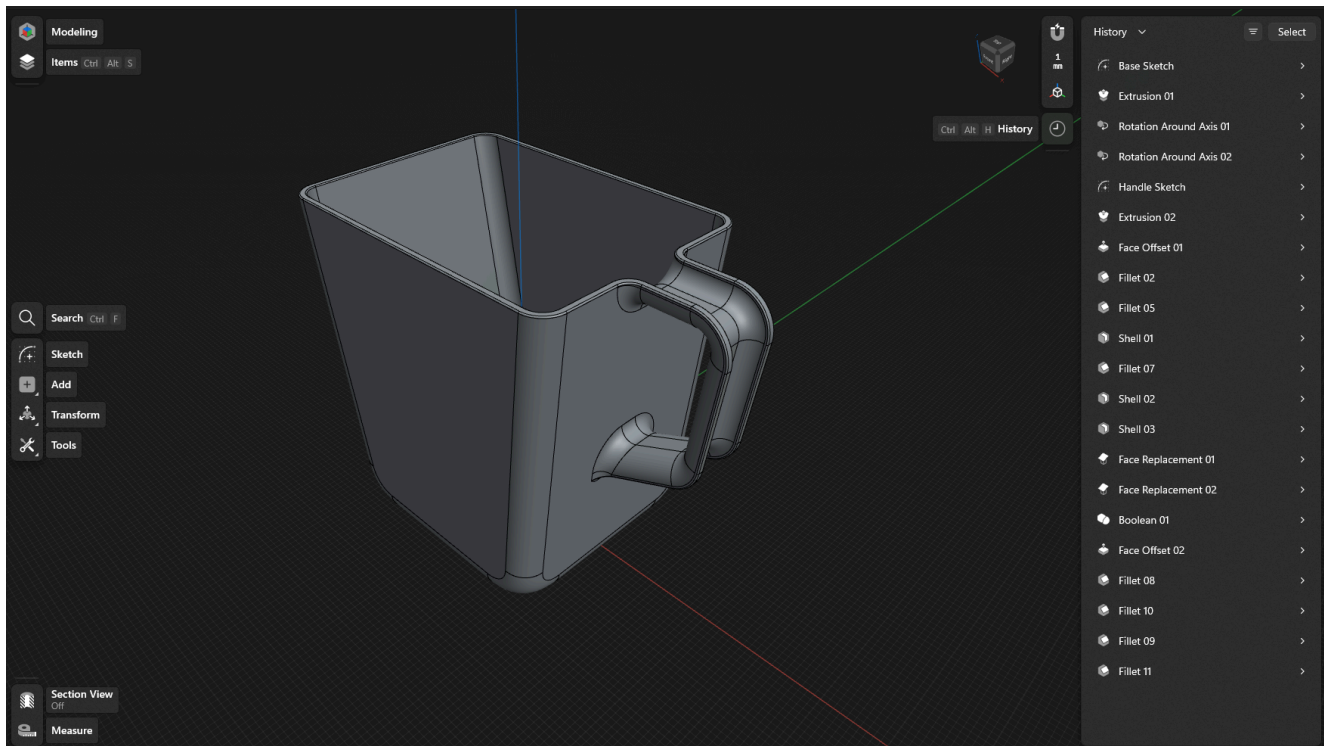
1. Open the expandable card for the first step with the first error message. In this case, we expanded **Extrusion 01**.
2. In the card, check the the description of the error message to see what's causing the problem. In this case, it states that we have 1 missing reference.
3. To view the missing reference, 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.

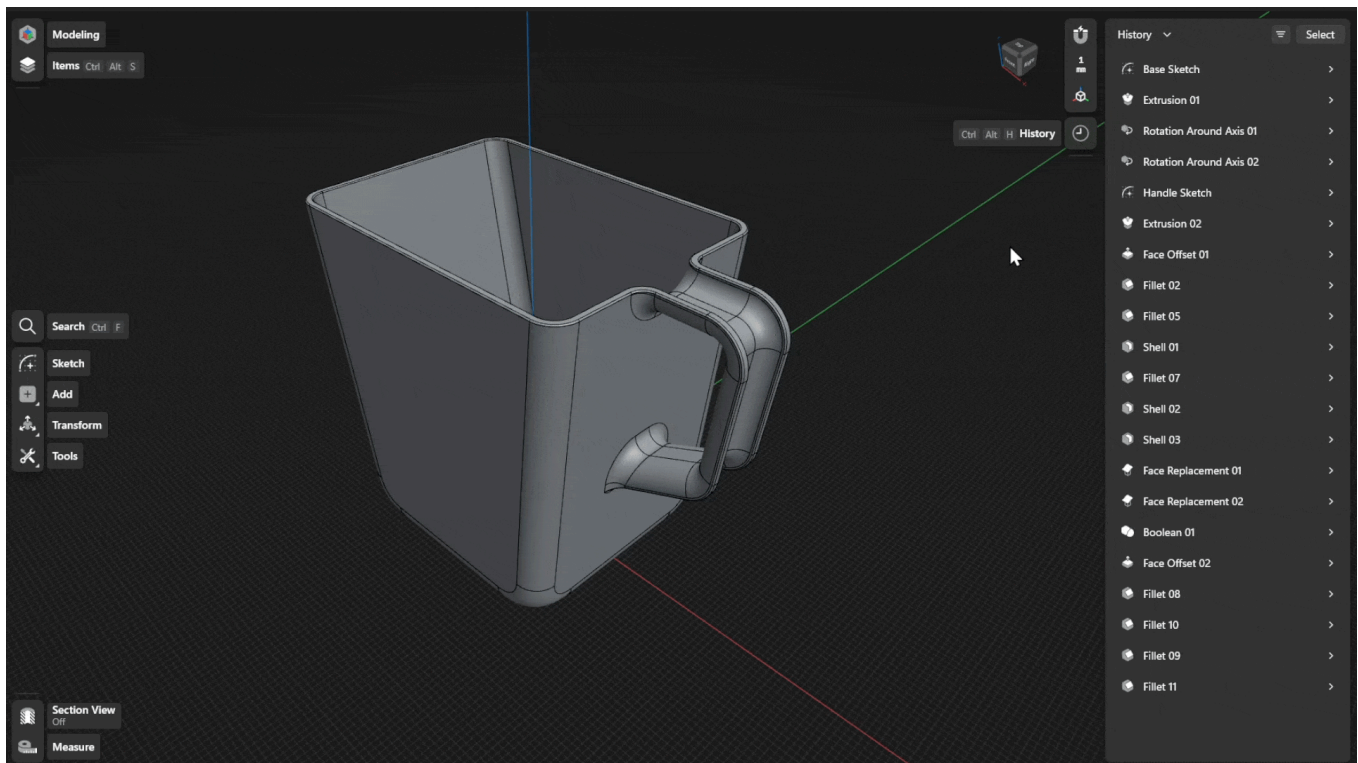
Example #2: Missing model parts due to a deleted sketch item

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

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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

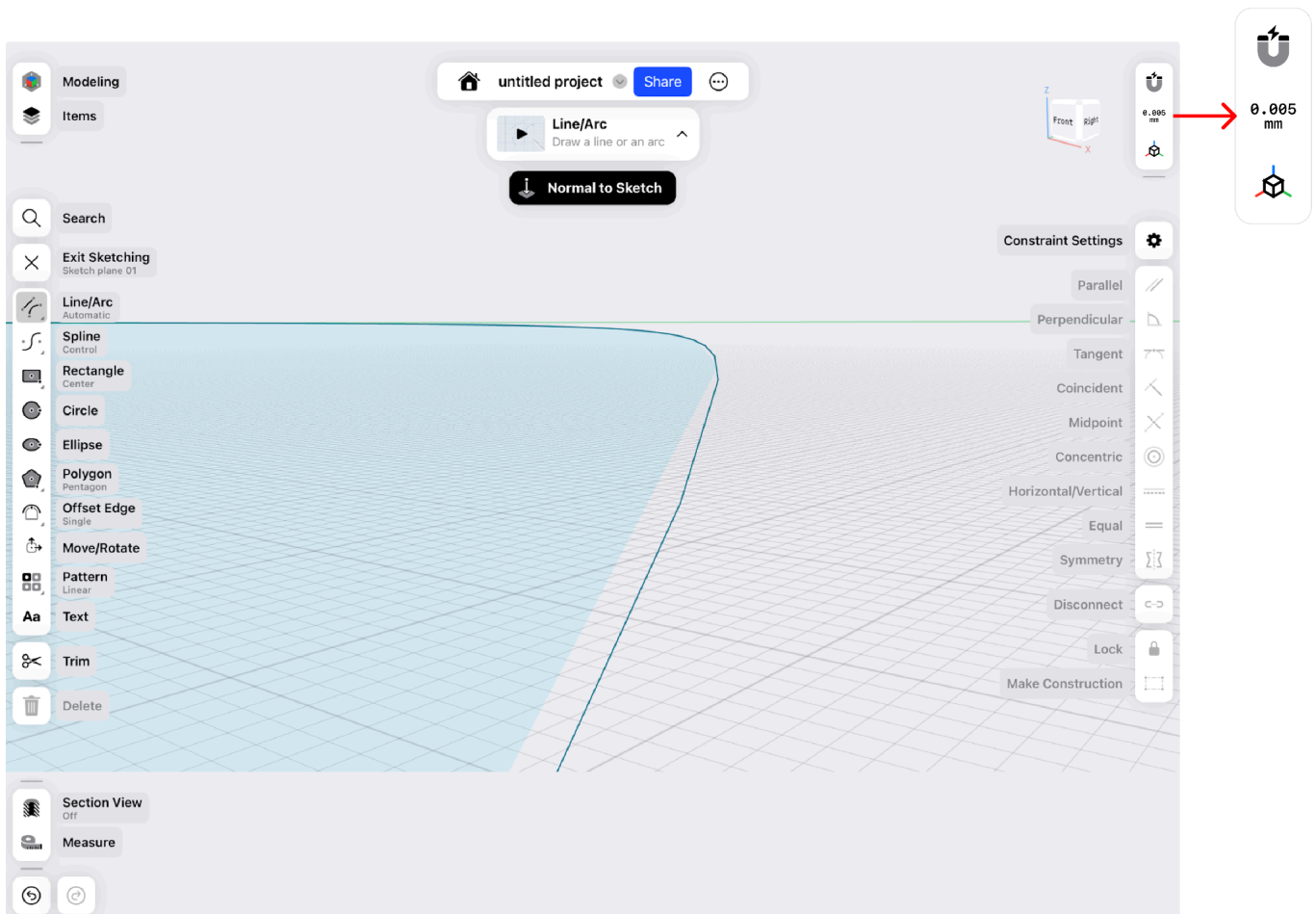
When zooming in on sketches, Shapr3D reduces the rendering quality to save on performance.

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

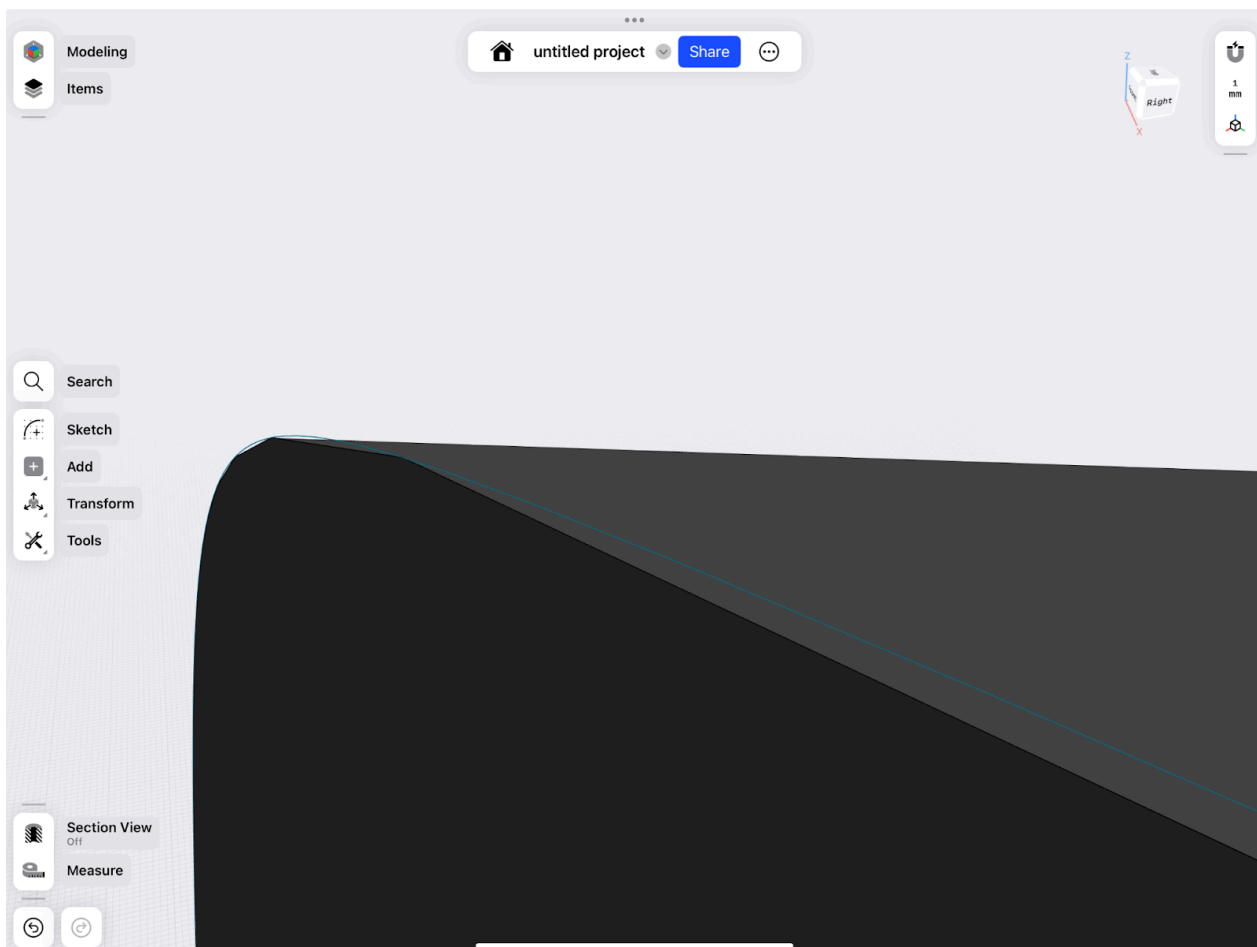
When zooming into the edges of bodies, Shapr3D reduces the rendering quality to save on performance. The reduced rendering quality doesn't affect the computational accuracy of the model.

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.

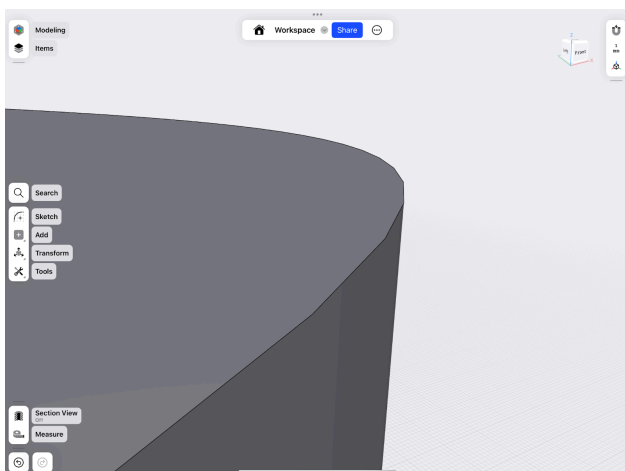
Curved edges appearing as jagged edges

Cause

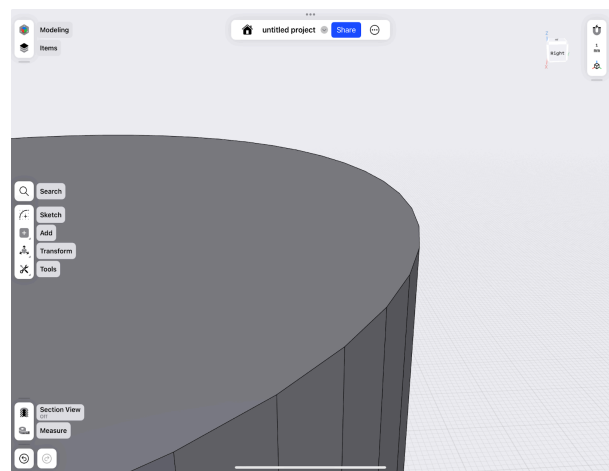
- When you have A few hundred bodies in your modeling space and you zoom in on edges, Shapr3D reduces the rendering quality to provide a smooth modeling experience.
- To save on performance, the detail of rendering may decrease depending on your level of zoom and the number of bodies in the modeling 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

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If you're having trouble resetting your Shapr3D account password or experiencing issues such as not receiving the email, try the troubleshooting steps below.

Need more help?

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

Send an email to support

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

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Shapr3D uses several third-party libraries, repositories, and other similar software packages. Thank you to the owners and contributors listed below.

- [3Dconnexion](#)
- [Alamofire](#)
- [Amplitude](#)
- [AppsFlyerFramework](#)
- [AWSCore](#)
- [AWSKinesis](#)
- [CleverTap-iOS-SDK](#)
- [CTNotificationService](#)
- [Dear ImGui](#)
- [Draco 3D Graphics Compression](#)
- [Eigen](#)
- [FBSDKCoreKit](#)
- [Filament](#)
- [Firebase/Core](#)
- [Firebase/Crashlytics](#)
- [Firebase/RemoteConfig](#)
- [Intel TBB](#)
- [Lib3mf](#)
- [LLVM](#)
- [Meshoptimizer](#)
- [MGSwipeTableCell](#)
- [MKStoreKit](#)
- [MikkTSpace](#)
- [Miniz](#)
- [moodycamel::ConcurrentQueue](#)
- [moodycamel::ReaderWriterQueue](#)
- [RapidJSON](#)
- [Reachability](#)
- [SDWebImage](#)
- [SQLite](#)
- [stb_image](#)
- [SwiftKeychainWrapper](#)
- [SwiftLint](#)
- [tinyobjloader](#)
- [USD](#)
- [Welder](#)
- [Xatlas](#)