Ψ M.A.R.S

Mechatronics and Robotics Society

MARS CAD Guide
Autodesk Inventor 2023

*

Contents

5

About This Guide

This Lookup Guide aims to provide a comprehensive summary of all the capabilities of Autodesk Inventor. It has been developed by UQ MARS with the intention of broadening the awareness around CAD software and serving as a manual for students while they they are developing their skills in 3D modelling. It is not intended as a resource to teach students how to use CAD, and it is recommended that students attend the offered CAD Workshop if they are completely new to the softwares. Instead, this is aimed towards students with an intermediate background in CAD and are looken to hone their skillset. This guide will aim to give a broad understanding of Autodek Inventor's various tools and in what applications they can be benefited from, and give greater awareness to powerful tools which are often underutilised. Note that, although this guide is specific to Inventor, most of the tools will also be present in other CAD software.

Using This Guide

Chapters are organised corresponding to the tabs of the workspace ribbon, and each one will cover a different core capability of Inventor. Over time, content will be added chapter by chapter. Within each chapter, sections are marked as either Green, Orange or Red. Green sections cover the more fundamental skills which are very important to master, while Orange sections have more niche use cases and Red sections cover advanced topics.

Because the length of this guide will only keep growing, we don't recommend that you try to remember everything that has been written. All key takeaways will be listed at the start of each chapter, and should provide a brief overview of the most important parts of the section. You can then refer to the rest of the chapter if you are interested in a particular feature or if you want to use it for a project and need more information.

We always welcome feedback and will keep updating our chapters for better coverage and readability. If you want a section to be clarified, expanded on or you find an error, you are welcome to provide feedback through Github Issues or via our discord at `https://discord.gg/JWzFrKeN`.

User Interface

Getting Around Inventor

Views and Layout

Collaborating and Content Libraries

Tools

Materials and Appearance

Document Settings

Measuring

Parameters

Document Style

Component and Assembly Properties

Work Features

2D Sketches

2D sketching is one of the most fundamental parts of Autodesk Inventor. A 2D sketch is essentially a collection of pieces of geometry, and are usually used as building blocks to create 3D features. Generally, when we create sketches we first place down any geometry that we want. We can then modify parts of the geometry and use constraints and dimensions to specify the position and size. We can then use the sketch to either analyse shapes or create 3D features.

This section will be a deeper dive into 2D sketching in Inventor, beyond just the basics of how to create sketches. We will highlight most of the auxiliary features that you can utilise to speed up your workflow or create more complex sketches. All of the features detailed in this chapter are accessible via the `Sketch` tab of the workspace ribbon, or via `Right Click` on the shared workspace. The ribbon is ordered into different tabs, which group together similar features and will be forming the basis for the subsections of this chapter.

2D Sketches: Key Takeaways

- Use Dimensions or Constraints to ensure your sketch is fully constrained. To make this faster, utilise snapping geometry to points and using inferred constraints.

- The Automatic Dimension tool will automatically create all the dimensions you could need.

- The bottom right of your workspace will display a lot of important information such as missing dimensions and coordinates.

- If you want to recreate a physical object, you can Import a Picture into your sketch and use your geometry tools (most commonly Splines - Interpolation Points) to draw over it.

- Project Geometry and Cut Edges are very powerful tools that lets you copy existing sketches and features into your sketch.

- Duplicate and pattern your shapes with Geometry Patterns.

- Change the Format of your geometry to make complex sketches easier to read.

- You can group pieces of geometry into Sketch Blocks to move around as one.

- Using Parameters allows you to quickly change the dimensions of sketches and objects, no matter where you are in the application.

- Inventor allows you to analyse the geometric properties of the sections of your sketches through Region Properties.

Creating a Sketch

2D sketches can be created in any plane or flat surface. There are many different ways to create a sketch, but the easiest by selecting 3D Model ▶ Sketch ▶ Start 2D Sketch and then clicking on the desired plane or flat surface.

Adding Geometry

Types of Geometry

There are several types of geometry which can be placed:

- Line: Creates either simple lines or joined segments of lines and arcs. Click on your starting point, and then your endpoint (to create a line) or click and drag your starting point (to create a curve). Finish the command by double clicking on the final endpoint or clicking `ESC` .

- Spline: Creates a spline (parametric curve), either by specifying points for the spline to pass through or control vertices.

- Equation Curve: Creates a curve based on a mathematical equation. The equation can be explicit or parametric, and use polar or cartesian coordinates.

- Bridge Curve: Creates a curve that joins two line/curve endpoints together such that the bridge curve is tangent to both segments (ref. Smooth Constraints).

- Circle: Creates a circle, using either a center point or three tangent lines.

- Ellipse: Creates an ellipse based on two axial lines

- Arc: Creates an arc, using the endpoints and either a center point or midpoint (Three Point Arc). Can also create an arc that joins a line's endpoint and is tangent to it.

- Rectangle: Creates a rectangle using various key points.

- Slot: Creates a slot or capsule, using various key points.

- Polygon: Creates a polygon of any order, and can be drawn inscribed or circumscribed.

- Fillets and Chamfers: Transforms sharp corners into curved corners (fillets) or bevelled corners (chamfers). The radius or angle of these fillets/chamfers can be adjusted in the popup dialog.

- Text: Creates text as geometry, which can be further extruded, swept, etc. Various formatting options can be adjusted in the popup dialog.

- Geometry Text: Creates text that lies along lines or simple curves. Cannot be used for more complex geometries such as splines.

- Point: Creates points.

Tips and Tricks for Adding Geometry

- You can usually specify the coordinates of key points or the dimensions of your geometry (magnitude and angle). Press Tab to specify coordinates or switch between multiple dimensions. You can change the coordinate type (for example, absolute vs relative) by `Right Click ▶ Coordinate Type` .

- The bottom right of your workspace will display a lot of useful information when creating geometry, such as coordinate positions, deltas, and dimensions.

- You can "snap" your geometry to existing points. A green dot will signify you are placing your geometry on an existing point, and a yellow dot will signify you are placing your geometry along an existing curve. This is often useful to make sure that all your lines connect to form a closed shape. If you are having trouble snapping to geometry, you can `Right Click ▶ Point Snaps` when placing down geometry and choose whether you want to snap to a midpoint, endpoint or various other points.

- The above is a case of constraint inference (ref. Constraints), and you can align your geometry in other ways, such as drawing a line which is tangent to another curve. You can temporarily disable constraint inference or point snapping by holding `CTRL` .

- When starting the Line command, you can easily create tangent or normal lines. Click your starting point on existing geometry, and then click and drag the starting point. The direction you drag determines whether the line will be tangential or normal.

- Splines are very powerful tools that are commonly used. If you have created a spline using control vertices, you can move these vertices around to change the shape of the spline. If you hvae created a spline using interpolation points, you can either move around the interpolation points or adjust the "handles" of the spline - these are construction lines that appear when you click on the spline, and represent the tangent lines at the interpolation points.

Projecting Geometry

The Project Geometry feature is a very powerful tool which allows you to copy existing geometry, whether 2D or 3D, to your current sketch. Try to always project geometry which is above the current sketch in the model browser. Generally, projected geometry is adaptive, which means changes to the reference geometry will be reflected in your sketch. To prevent this, hold down `CTRL` when projecting geometry, right click on the geometry and select `Break Link` , or manually delete the Project Geometry Constraint (ref. Constraints). There are the following options:

- Project Geometry: Projects 2D sketches, 3D sketches, object edges and more into your sketch.

- Project Cut Edges: Projects a 3D object or surface as if the sketch was to cut through the cross-sectin of it.

- Project Flat Pattern: Used predominately in sheet metal projects, unfolds an object into the plane.

- Project to 3D Sketch: Used to project your 2D sketch onto a 3D surface. For more information on this group of features, see 3D Sketches or 3D Model ▶ Surface.

- Project DWG Geometry: Allows you to copy a DWG (drawing) file from your workspace into a sketch. Not very useful for most scenarios.

Geometry Patterns

Inventor allows you to duplicate geometries to streamline your work. Geometry can be mirrored or repeated in a circular or rectangular pattern. The patterns are associative, so changes to any one of the object instances will change the entire pattern.

Modifying Existing Geometry

The Modify tab allows you to make changes to geometry you have already created. You can:

- Move geometry

- Copy geometry to a new location

- Rotate geometry

- Scale geometry

- Trim parts of your geometry off by points of intersection

- Extend a line or other geometry until it intersects with another geometry

- Split geometry into multiple parts according to points of intersection.

- Stretch certain geometry out

- Offset geometry by creating a slightly larger or smaller copy.

Tips and Tricks

- You can change how Inventor treats dimensions and constraints (i.e if it breaks constraints or throws an error) by clicking the ≫ button on the popup dialog of the modify tools and changing the settings.

- By checking the Precise Input box on the popup dialog of many of these tools, you can specify the exact coordinates with which you want to modify your geometry with.

- You can choose whether an offset geometry adapts to changes in the original geometry by right clicking when using the Offset tool and toggling Constrain Offset . Under these options, you can also change whether Inventor offsets entire loops or single line/curve segments.

Constraints and Dimensions

Constraints and dimensions determine the relationships between geometries. If a sketch is not fully constrained, it means that you can freely move it around by simply clicking and dragging. We recommend that, in general, you fully constrain all sketches. The bottom right of your workspace will tell you how many more dimensions/constraints you would need to achieve this.

Dimensions

Dimensions allow to you to display and specify the length or angle between two pieces of geometry. Dimensions can also be added when creating geometry, by pressing `Tab` to toggle between possible dimensions. The following dimensions can be created (noting that curves are usually dimensioned by the center point):

1. The distance or angle between two lines

2. The distance between two curves/points. Drag the cursor to specify if this distance is vertical, horizonal or euclidean (by clicking between two points).

3. A line and curve/point: the distance between them in the direction orthogonal to that of the line

Two types of dimensions can be created: regular dimensions and driven dimensions. Driven dimensions are dimensions which are redundant (already specified by existing constraints/dimensions), and thus are only used to display measurements. They are signified by brackets. On the other hand, regular dimensions can be edited to change the geometry. Regular dimensions can be made and unmade driven with `Format ▶ Driven Dimension` or `Right Click ▶ Driven Dimension` .

Dimensions can be edited by double clicking on the dimension. The formatting of the dimension can be modified by `Right Click ▶ Dimension Properties` , allowing you to specify naming, precision, and tolerance type for both the individual dimension and the default settings.

Types of Constraints

Constraints allow you to set relationships between two pieces of geometry. The types of constraints are:

1. Coincident: a point is constrained to another point or along another curve

2. Collinear: two straight lines (or axes) constrained to lie along the same line

3. Concentric: two curves, ellipses or circles constrained to the same center point

4. Fixed: locks geometry in place

5. Parallel: two lines constrained to lie parallel

6. Perpendicular: two lines constrained to lie perpendicular

7. Horizontal: a line or pair of points constrained parallel to the x axis

8. Vertical: a line or pair of points constrained parallel to the z axis
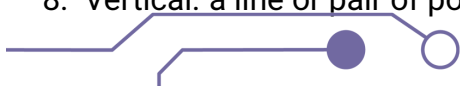
9. Tangent: two curves constrained to be tangent to each other

10. Smooth: a splined curve constrained to be tangent to and join endpoints with a curve

11. Symmetric: two curves constrained to be symmetric about a third selected line

12. Equal: two curves constrained to have the same radius or two lines constrained to have the same length

13. Project Geometry: essentially a fixed constraint, it cannot be added to geometry but instead is automatically applied to any projected geometry. If you want to edit projected geometry, you need to remove these constraints.

Adding Constraints

Constraints can be added through the `Constrain` tab. Select the desired constraint in the tab and then select the geometry to be constrained. For some constraints such as the fixed or equal constraint, you can select multiple geometry and then click on the desired constraint to apply to all.

Viewing and Editing Constraints

All constraints can be viewed by right clicking on the workspace and selecting `Show All Constraints` or using the `F8` hotkey. Constraints can be hidden by right clicking on the workspace and selecting `Hide All Constraints` or using the `F9` hotkey.

To toggle the visibility of a selected geometry's constraints, click on the `Show Constraints` icon under the `Constrain` tab in the ribbon.

To delete all constraints from geometry, select the desired geometry, press right click and select `Delete Constraints`.

To delete specific constraints, show constraints either by using F8 or selecting the parent geometry, click on the respective constraint icon beside the geometry and press DEL.

Constraint Inference, Relax Mode and Settings

Sometimes constraints can be automatically added while placing geometry. For example, drawing a line which is horizontal will create an inferred horizontal constraint. If you find that a constraint is being inferred with the wrong geometry, you can first mouseover ("scrub") the desired geometry. You can temporarily disable inferred constraints by holding `CTRL` when placing geometry.

Inferred constraints can be persistent or non-persistent. Persistant constraints remain after the geometry is placed, however non-persistent constraints will not, and will only help you while placing geometry. In general, inferred constraints will appear as a small grey box when you are creating geometry. They will also either include yellow/green points (for coincident constraints) or solid lines (for other constraints). Any dotted lines that appear are NOT inferred constraints and are merely guide lines that will not persist (though otherwise they behave similarly).

Usually, constraints will restrict you from freemoving geometry. For example, if you have two circles with a concentric constraint, moving one circle will also cause the other to move. When Relax Mode is enabled, geometry can be moved freely. Instead of also moving constrained geometry, any constraints which are broken will be deleted. In the above case, only the selected circle will move and the concentric constraint will be removed. You can also choose to only enable Relax Mode for specific constraint types.

Settings for constraints can be modified under `Constrain ▶ Constraint Settings` You can:

1. Under `Inference` , turn all inferred constraints on/off

2. Under `Inference` , turn specific inferred constraints on/off, such as turning off any inferred horizontal constraints

3. Under `Inference` , choose to infer parallel/perpendicular constraints over horizontal / vertical constraints, when there is an option

4. Enable Relax Mode

You can also edit the scope at which constraints are inferred. Under

`Constrain ▶ Constraint Inference Scope` , the three options are:

1. Geometry in current command: constraints will only be inferred with the same type of geometry (e.g line, arc, rectangle). Remember that guide lines will still appear with reference to other geometry types.

2. All geometry: constraints will be inferred with all geometry, regardless of type or location.

3. Select: you can choose which pieces of geometry constraints can be inferred with.

   It is also possible to realign a coordinate system, either by relocating the origin or the two axes. Select `Constrain ▶ Edit Coordinate System` , and then relocate either the origin by clicking on a new vertex or the axis/axes by clicking on new edges. To flip the direction of the axes, right click and select the appropriate option. You may need to move the camera in order to do this, and remember that your sketch needs to be below any reference objects in the model browser.

Automatic Dimensions

By selecting `Constrain ▶ Automatic Dimensions and Constraints` , dimensions and constraints can be added to fully paramaterise the sketch. This is a very powerful tool!

Formatting

The graphic display of geometry can be adjusted under $\boxed{\text{Format}}$. Construction formatting identifies geometry as being a scaffold to the sketch and not part of the final product, such as radius lines. These are still visible and can be dimensioned/constrained, however cannot be used for extrusions or any other 3D feature. Some shapes, such as center point rectangles, automatically come with construction lines, and in order to use these lines for 3D features it must be disabled in the same way construction lines are enabled.

Centerline formatting signifies a line being a symmetric center of geometry, and behaves in the same way as construction lines. Center points have the same philosophy.

Geometry can also be custom formatted by changing the pattern, colour and weight. These custom formats can be turned on/off by clicking $\boxed{\text{Format} \blacktriangleright \text{Show Format}}$.

Layout

Assembly and Component Layouts

This is part of a more complex feature of Inventor which comprises of creating hierarchies of assemblies, components and parts. In short, the `Make Part` and `Make Components` features allows you to import 2D sketches into assemblies as layout parts. This is useful because you can create 3D features in the assembly without changing the layout file, and as such these files can be used concurrently for multiple projects. This will be covered in a later section and is not a necessary tool to know.

Sketch Blocks

Sketch blocks are similar to object grouping in Microsoft Office; it allows you to constrain portions of sketches to be fixed relative to each other, and instead of moving each piece of individual geometry the entire block moves. This is especially useful if you copy and paste a lot of sketch data from other applications or do a lot of sketching over images - it removes the need to constrain every line and curve. Sketch blocks can be created under `Layout ▶ Create Block`

Insert

Inserting Images

Images can be inserted into a 2D sketch under `Insert ▶ Image` , and behave like construction-formatted rectangles in terms of geometry. Inserting images is a fantastic way to model objects that do not have CAD files, as you can use them to freehand outlines - learn how to create 3D models from just 2D profile images in the 3D Sketch section!

Inserting Points Spreadsheets

If you have a large number of points that you want to copy into modelling software - such as from point cloud scanners - your best option is to import it via an excel spreadsheet. Your first row should contain only your units specification, your second row should contain the headings x, y and (optionally) z. Each subsequent row should represent the coordinates of one point.

Inserting AutoCAD Files

If you use AutoCAD to create drawings, you can import 2D drawings into a sketch here. This should work for any DWG or DXF file.

Managing 2D Sketches

After creating a sketch, there are several things you can do:

Right Click on the sketch under the part navigator and you can:

- Redefine: if you want to move a sketch from one plane to another. To see the current sketch plane, select Show Input .

- To change the appearance properties of the sketch, such as colour and line style.

- Engineer's Notebook: The Engineer's Notebook allows you to log your progress through the design stage. You can add a note in your notebook about your sketch by selecting Create Note .

- Export: You can export your sketch as a drawing format.

- Visibility: You can toggle the visibility of the entire sketch and the sketch dimensions.

Parameters

The majority of information about parameters will be covered in a later section, however they are very commonly used for sketches. With parameters, instead of specifying numeric values for dimensions, you can instead assign a global variable to it, which you can change later at will. This is useful if you have multiple dimensions which reference the same physical measurement, or if you want to create a series of the same object but with different sizings - such as for screws. Two useful things you can do with parameters are:

- Link your parameters with an excel file that you can adjust in real time

- Import and export parameters from XML files

- Share parameters between files

More Sketch Options

While still in your sketch, you can $\boxed{\text{Right Click}}$ on the workspace and:

- Degrees of Freedom: Wither on the workspace or for a specific piece of geometry, you can display the degrees of freedom present. This is useful to make sure that you have enough dimensions - if you don't, you will see which directions have not been constrained yet.

- Slice Graphics / Hide Others: If you find that other pieces sketches and 3D features are being obstructive, you can either completely isolate the sketch by selecting $\boxed{\text{Hide Others}}$ or cut through the existing 3D features by selecting $\boxed{\text{Slice Graphics}}$.

- By selecting $\boxed{\text{Region Properties}}$, you can analyse the important geometric features of an area in your sketch, including moments of inertia and radii of gyration.

- Curvature Lines: if you want to project the radial and angular lines of a curve outwards, you can right click on the curve and select $\boxed{\text{Display Curvature}}$. To change the settings of these lines, select $\boxed{\text{Setup Curvature Display}}$.

3D Sketches

2D vs 3D Sketches

Drawing 3D Sketches

Projecting and Combining 3D Sketches

3D Constraints

Modifying 3D Sketches

3D Modelling

This is where all the magic happens! These tools transform 2D sketches into 3D features, surfaces and more. Is it integral for any 3D modeller to understand how to effectively utilise the many tools at their disposal. There will usually be many ways to model an object, but good practices dictate to do it in a way that is the most robust and flexible.

This section will outline the 3D modelling tools which are available in Inventor, as well as when they should be used and the meanings of the numerous options each one is equipped with. In addition, the best practices of modelling will be given.

Creating Features

Types of Features: At a Glance

- Extrude: takes a cross-section from a 2D sketch and extrudes it into a prism.
- Revolve: takes a section from a 2D sketch and rotates it around a specified axis.
- Sweep: takes a cross-section from a 2D sketch and pulls it along a 2D/3D path to create a pipe-like solid.
- Loft: creates an object that transitions between multiple cross-sections.
- Coil: takes a profile and transforms it into a helical, spiraled or threaded object around a specified axis.
- Emboss: takes text or other geometry and engraves or embosses it onto an object.
- Decal: wraps an image to a face or multiple faces.
- Rib: adds ribbed or webbed support walls to an object.
- Import: allows you to import other CAD formats into your Modelling.
- Unwrap: unwraps a solid body, often of sheet metal, until it becomes flat.
- Derive: allows you to import an Inventor model to form the base of the current part, which you can then build on top of and is adaptive.

Examples of When to use Features

- Extrude: you want to create basic objects, prisms and boxes.
- Revolve: you have a solid of revolution that you would like to model, such as a sphere or torus.
- Sweep: you would like to design an intricate slide, so you take the cross-section of the slide and sweep it around your 3D path.
- Loft: you identify using stress analysis two major load areas in a solid body, one large area and one small area. You would like to add structure between these areas to reinforce it, so you use a loft to create a solid that transitions from the large area to small area.
- Coil: you would like to manually model a spring, threaded screw or spiral.
- Emboss: you would like to engrave your brand name and logo into your products.
- Decal: you have a vehicle or other product that would be manufactured with an image printed on it.
- Rib: you have a corner that you would like to reinforce by adding a diagonal brace.
- Import: you have a model in a different format but want to use it as in Inventor part.
- Unwrap: you have a sheet metal object and want to unwrap it to give you a clear idea of how to manufacture the part prior to applying bends.

- Derive: you have a 3D modelled product and want to personalise it by adding an engraving. Insert the generic part with the Derive tool, and then add an your engraving. Changes to the generic part will update in the new part, but adding an engraving in the new part will not add it to the old part.

Common Options

When using the Create tools above, most of them have options which allow you even more flexibility with 3D modelling:

- Surface Mode: Instead of creating a solid object, you can instead create a surface. For example, instead of a sweep creating a solid pipe it will create a hollow one.

- Profiles: the cross-sections you want to be turned into 3D features. You can often choose multiple, and they don't always have to be in the same sketch, plane or even the same orientation.

- Presets: if the Create tool has many different parameters, you can create presets to speed up your workflow

- Direction: direction can either be default, flipped, symmetric (same distance in each direction) or asymmetric (different distance in each direction)

- Solid body: a part can be comprised of several distinct bodies. When creating a feature, you can choose which solid body it will be incorporated into.

- Output (Boolean): the type of operation being performed. You can join the 3D feature to an existing solid body, cut the feature away from the body, take the intersection between the feature and body or create an entirely new solid.

- Numerical Values: when entering numerical values, you can often click the arrow beside the input box to quickly measure dimensions, access recently used values or tell Inventor to use an existing dimension as a reference (updating the dimension in the future will also update the feature).

- Keep Sketch Visible: under the hamburger menu on the top right of the popup dialog, you can specify to keep the sketch visible after executing the feature. This allows it to be reused.

Additional Options: Extrusions

- From (Start Plane): you can choose the plane the extrusion will start at - it does not have to be the sketch plane!

- Distance (End Plane): you can specify how far the profile extrudes by:
  - Manual Distance: specify the numerical distance of extrusion
  - Through All: the extrusion will pass through all objects (for cut or intersect extrusions only).
  - To: select the plane or feature that the extrusion will end at. For curved planes, Alternate Solution specifies to choose the closest side of the curved surface.

- To Next: the extrusion will end at the next feature or plane. Said feature or plane should entirely overlay the cross-section profile.
- Taper: the extrusion tapers slightly inwards or outwards

Additional Options: Revolve

- From (Start Plane): you can choose the plane the revolution will start at - it does not have to be the sketch plane!
- Distance (End Plane): you can specify how far the profile revolves by:
  - Manual Angle: specify the numerical angle of revolution.
  - Full: a 360 degree revolution.
  - To: select the plane or feature that the extrusion will end at. For curved planes, Minimum Solution specifies to choose the closest side.
  - To Next: the extrusion will end at the next feature or plane. Said feature or plane should entirely overlay the cross-section profile.

Additional Options: Sweep

- Path: Choose the path (a 2D/3D curve or edge) for the profiles to follow. It does not necessarily have to intersect the profiles, but the path must intersect the profile plane.
- Behaviour: Determines how the profile follows the path:
  - Follow Path: the profile is swept along the path, where every cross section taken normal to the path stays constant. Adjust the twist and taper as necessary - 360 degree twist corresponds to one single twist, and positive/negative taper corresponds to a widening/narrowing of the sweep away from the start point.
  - Fixed: the profile is swept along the path, where the cross section taken parallel to profile plane stays constant. Twist and taper are not available.
  - Guide: the profile is swept along a path in the same manner as Follow Path. A second guide rail is used to determine how the profile scales (either in one or both directions) along the path - it guides the outside of the sweep.

Additional Options: Loft

- Sections: the cross-sections for the loft to transition through. You need at least two profiles, however they do not have to be in parallel planes. You can also loft between two profiles on the same plane, which will create a flat (2D) feature. You can also loft to single points rather than sections by simply selecting a point.
- Rails: the paths for the loft to follow
- Behaviour: how the rails are treated - either as outer rails or center lines.

- Conditions: allows you to control what angle the loft meets the sections at (i.e tangent to the plane)

- Closed Loops: joins the last section to the first section to form a closed loop object.

- Merge Tangent Faces: blends together faces that end up tangent with each other.

- Transition:

Additional Options: Coil

- Axis: the axis the sketch will coil about. An internal axis will create a threaded object, and an external axis will create a helical object.

- Behaviour: controls the physical properties of the coil, such as pitch height and revolution. Allows you to create a spiral (no vertical displacement)

Additional Options: Emboss

- Behaviour: determines whether the emboss raises up or recesses into the object. There is a third option, Emboss/Engrave From Plane, which allows you to create an emboss/engraving where the depth changes. In this option, you can also transition from an emboss to an engraving, which is a very powerful tool.

- Taper: allows you to determine how the profile tapers (scales) when Emboss/Engrave From Plane is enabled.

- Appearance: allows you to automatically change the visual appearance of the emboss

- Wrap to Face: if embossing on a curved face, allows the emboss to follow the curve around. With this enabled, the emboss will always have the same depth at every location.

Additional Options: Decal

- Automatic Face Chain: the image automatically wraps over adjacent faces.

Additional Options: Derive

- Derive Style: determines how the solid bodies of the derived part import.

- Status: allows you to select which elements of the part will be imported.

- Options: more options which, most importantly, allows you to scale and flip the part before importing.

Additional Options: Import

- Import Options: some file types have additional options (beside the "Open" button) which allows you to control the unit imported, change the subformat and more.

Additional Options: Rib

- Style: allows you to specify whether the ribs are created parallel or normal to the sketch plane. This is not important for basic ribs, but further customisation usually requires the style to be Normal to Sketch Plane

- Thickness: determines the thickness of the ribs. The direction this thickness refers to depends on the style.

- Hold Thickness (Normal to Sketch Plane ¿ Draft): specifies whether the quoted thickness is taken from the top or bottom of the taper

- Draft Angle (Normal to Sketch Plane ¿ Draft): adds a taper to the normal face of the rib.

- Boss: allows you to create ribs for bosses (which are like standoffs and usually have ribs on both sides).

- Distance: determines the thickness of the ribs in the other direction.

Additional Options: Unwrap

- Auto Face Chain
- Alignment
- Behaviour

Primitives

Primitives provide shortcuts to easily create basic features: a box, sphere, cylinder or torus. These features just reuse the Create features detailed above.

Modifying 3D Features

You can apply the following modifiers to change already existing 3D features. It is often better practice to use one of these tools, such as the Hole feature, instead of incorporating a hole into your sketch for the 3D features.

Modifiers: At a Glance

- Hole: allows you to create holes of all kinds
- Fillet: rounds out sharp edges and corners
- Chamfer: applies a bevel to sharp edges and corners
- Shell: hollows out an object
- Draft: angles existing faces
- Thread: adds a thread to faces (including holes)
- Combine: takes two solid bodies and joins, cuts or intersects them (the same options as 3D Create Features)
- Thicken/Offset: thickens or thins faces
- Split: Splits solids and faces apart along planes, sketches or surfaces
- Mark: adds text and other markings to faces. In essence, this splits the face and uses the border lines to show the marking
- Delete Face: removes selected faces, leaving the other faces on the 3D feature intact
- Move Bodies: allows you to move around solid bodies in relation to each other
- Bend Part: takes an object and bends it over a line
- Copy Object: duplicates objects
- Direct Edit: allows you to edit imported models

Hole

Holes can be created at points that have been drawn on 2D/3D sketches. Alternatively, by selecting the Allow Center Point Creation option under Input Geometry, points can be added freehand. They have the following options:

Hole Type: Holes can be regular, tapped (threaded), tapered or a clearance hole, depending on the utility required. To design to fit a certain type of fastener, screw standards and sizes can specified under the Fastener tab.

Seat Type: The hole can have a countersunk, counterbore/spotface or no seat.

Behaviour: Determines the direction of hole, when or at what distance the hole terminates, and the geometry of any tapers or seats. Under Advanced Options Extend Start, the hole can also be extended so it extrudes both directions.

Fillet

Selection Priority:

Fillet Body:

Fillet Radius:

Edge Sets:

Edge Rolling:

Face Chaining:

Patterning 3D Features

In the same manner as 2D sketches, most 3D features can be patterned to streamline work. Mirror features about a plane, or repeat it in a rectangular or circular pattern. Patterns are associative, so changes made to one object reflect to the others. Follows the same principles as 2D patterns, however there are a few more additional options which are available:

Pattern Orientation

Choose whether the pattern copies stay in a fixed orientation or become rotated. For a circular pattern, this dictates whether all object face the same cartesian direction or all face inwards. For a rectangular pattern, determines whether all objects face the same cartesian direction or all lie tangent to either the first or second direction (this is under the additional options).

Path-Based Patterns

Unlike 2D sketches, rectangular patterns can also be generated along paths, including curved paths. Simply choose a custom 2D/3D sketch (or the edge of a 3D feature) and select it as the direction of patterning.

Sketch-Driven Patterns

Inventor also allows you to place down duplicate features according to a sketch of points. The feature is duplicated for each point, with the base point of the feature being the point that maps on top of each sketch point. The reference faces dictate the orientation (the face for the duplicates to lie tangent to). These can both be adjusted.

Creation Method

This defines the method Inventor uses to generate duplicates, and although this is usually unimportant certain situations require certain creation methods.

Optimised: This is the fastest method, so useful for large quantities of duplication. However, it does not permit overlapping duplicates or patterns that intersect with other faces. Under the hood, it duplicates the faces of the features.

Identical: The second fastest method, for when optimised is not possible. Under the hood, it duplicates the entire feature.

Adjust: The slowest method, with each duplication have potential adjustments made to it. Used for the preservation of design intent and constraints - where it is not needed to keep each instance exactly the same. The best use-case for this is for a part which has been extruded to "terminate at face". Each duplicate will also be set to the same termination condition, even if it results in objects being lengthened and shortened from the reference feature.
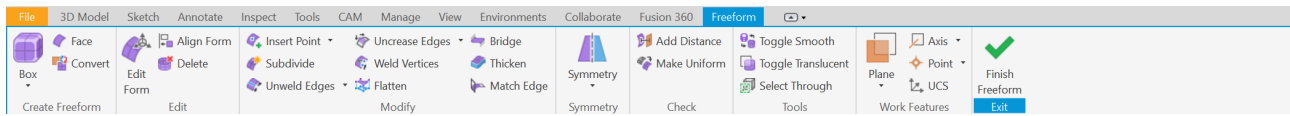
Freeform 3D Modelling

Freeform modelling is an alternative technique to 3D modelling, which lets us directly manipulate objects like they are plasticine. This style of modelling is more similar to the mesh-based design you may use in programs such as Blender, and although it can take practice to get used to, it is a much faster way of creating complex 3D shapes and surfaces. In freeforms, you deal with meshes - surfaces divided into many grids. This surface may wrap back into itself to form a 3D object, but freeforms will always be hollow; there is no mass to it.

This section will cover how to create freeforms and the various tools used to manipulate them. It will also give you tips on when to best use freeforms and the design process associated with them.
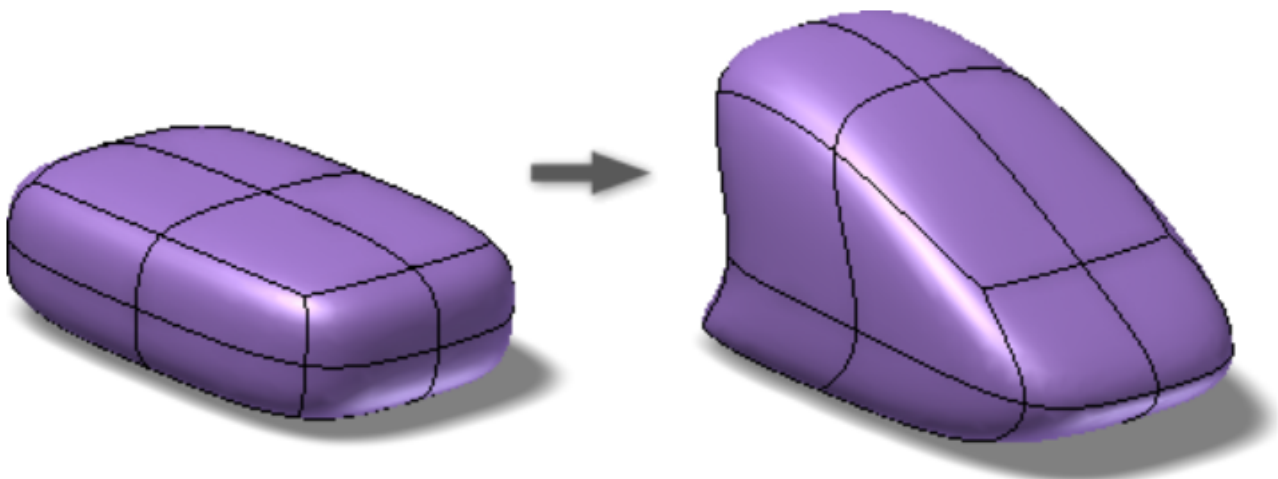
Creating Freeform Models

We can create freeforms from the 3D Model → Create Freeform panel. This will then put us in the Freeform environment, allowing us to create additional freeforms and edit them. Typically, we start our freeform model with one 6 primitve shapes, depending on which one best resembles the final shape we want our freeform to take. These base shapes are as follows: box, plane, cylinder, sphere, torus & quadball. Note that for spherical or revolved shapes, we usually prefer quadball to sphere because it deforms uniformly in every direction.

If we want a more complex shape, we can also start a freeform from a face (a polygon-shaped plane) or base it off an existing 3D model or surface. For more complex freeforms that need to be integrated with regular 3D modelling, this is usually the way to go, because you can exactly define how the freeform will look initially.

When creating a freeform, we can customise it in several ways. For primitive shapes, we can define the size of each dimension, or use the visual arrows to manipulate the shape as we

see fit. We can also define if we want to enforce symmetric constraints in any direction (see Symmetry & Mirroring). Finally, we can specify the number of faces our freeform is split up into in each direction. This controls how fine our mesh is, and therefore how much detail we can add.

Once we've created our freeform, the Freeform ribbon will open. We can additional freeform objects in the one instance under ` Freeform → Create Freeform ` . Usually we add planes to cover up holes in our model, or add additional bodies that we will later join to form one body. Never use the same freeform instance to create multiple parts; you should create each part separately.

Visual Tools

There are four visual options available specifically for freeform modelling.

` Freeform → Check → Add Distance ` :
Adds a constrained dimension between a feature of a freeform body and a plane. This is useful for Model-Based Definition paradigms. Note that this has to be done in Smooth Mode rather than Blocky Mode.

` Freeform → Tools → Toggle Translucent ` :
Change the opacity of the model.

` Freeform → Tools → Select Through ` :
Enabling this option will allow you to highlight and select geometry that is behind other geometry. This option is initially disabled, meaning that only parts of the model you can see will be selectable. Toggle this if you want to select all geometry or only geometry visible from a certain plane.

` Freeform → Tools → Toggle Smooth ` :
Toggle between the smooth freeform model and it's blocky representation. It is usually better practice to model in blocky mode, as it shows the underlying geometry and is easier to edit. Smooth mode is usually the final product, so is reserved for renders and the like. This is the "actual" shape of the freeform.
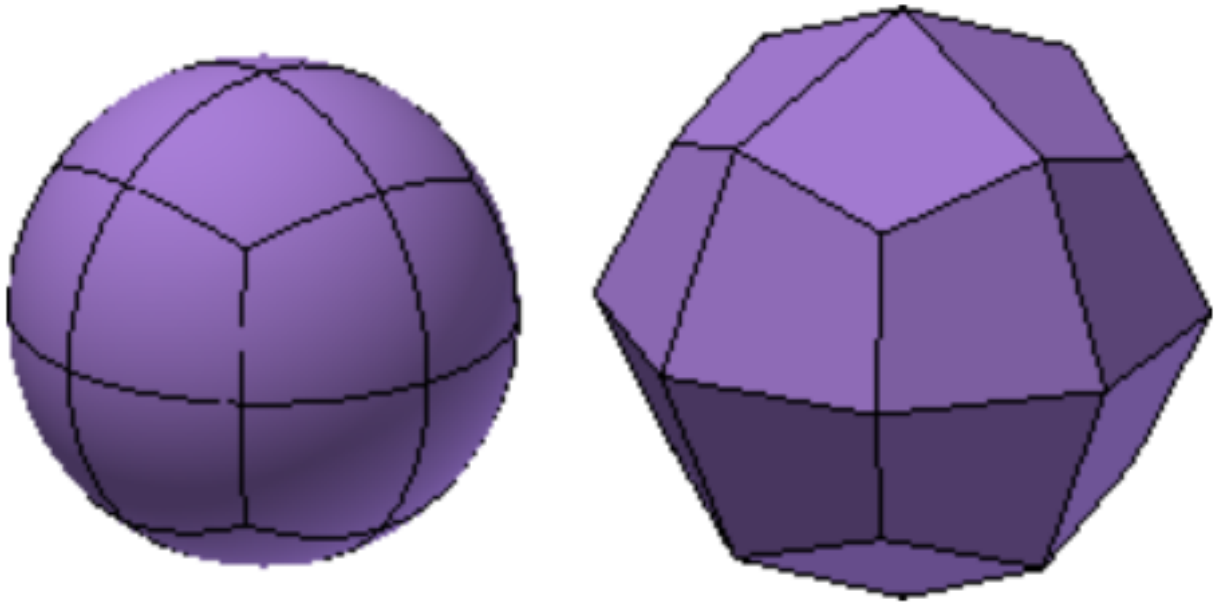
Editing Freeform Models

After creating a freeform body, we can change its geometry with ⟨Freeform → Edit → Edit Form⟩
. Select one more multiple bodies, faces, edges or points - it is convenient to drag select an
area of the body and then using the filtering options to select only faces, only edges or only
points. (Double clicking on an edge will instead select the edge loop it belongs to). These
geometries can then be translated, rotated, and/or scaled in all 3 dimensions, morphing the
surrounding geometry as needed. We can either specify these transformations in the dialog
box or use the visual arrows and points. Take note of the undo, redo and reset buttons in the
dialog box. When editing freeforms, also have options for Extrude Mode, Soft Modifications
and Transformation Coordinates.

Make sure to be in blocky mode when editing models. If you're in smooth mode, it's near-
impossible to be able to predict how changes affect the overall shape. The blocky represen-
tation makes it very clear what is being changed.

The Nuances of the Extrude Mode

⟨Extrude⟩ is an option that instead of morphing the surrounding geometry will only change the
highlighted geometry. This is most commonly used for faces. Typically in transformations,
pulling a face up will also pull the surrounding faces up to maintain smoothness. In extrude
mode this is not the case, and it will look like a regular extrusion. This creates sharp angles
in the blocky model, which translates to concave curves.

A useful feature of extrusions is to create shells. Extrusions can be cut into the existing
geometry to create hollow sections, which can then lead to very interesting concave shapes.
A common technique is to extrude a face in one direction and then immediately the other,
which removes the solid mass but keeps the outer shell.

Extrusions also have some quirks when used with edges. In most cases, translating will
not change its orientation (i.e there will be no rotation). However, after selecting the Extrude

mode when making edge transformations, the resulting edge will associate with surrounding edges. Trying to transform this new edge will then pull along other edges in ways that were not previously possible.
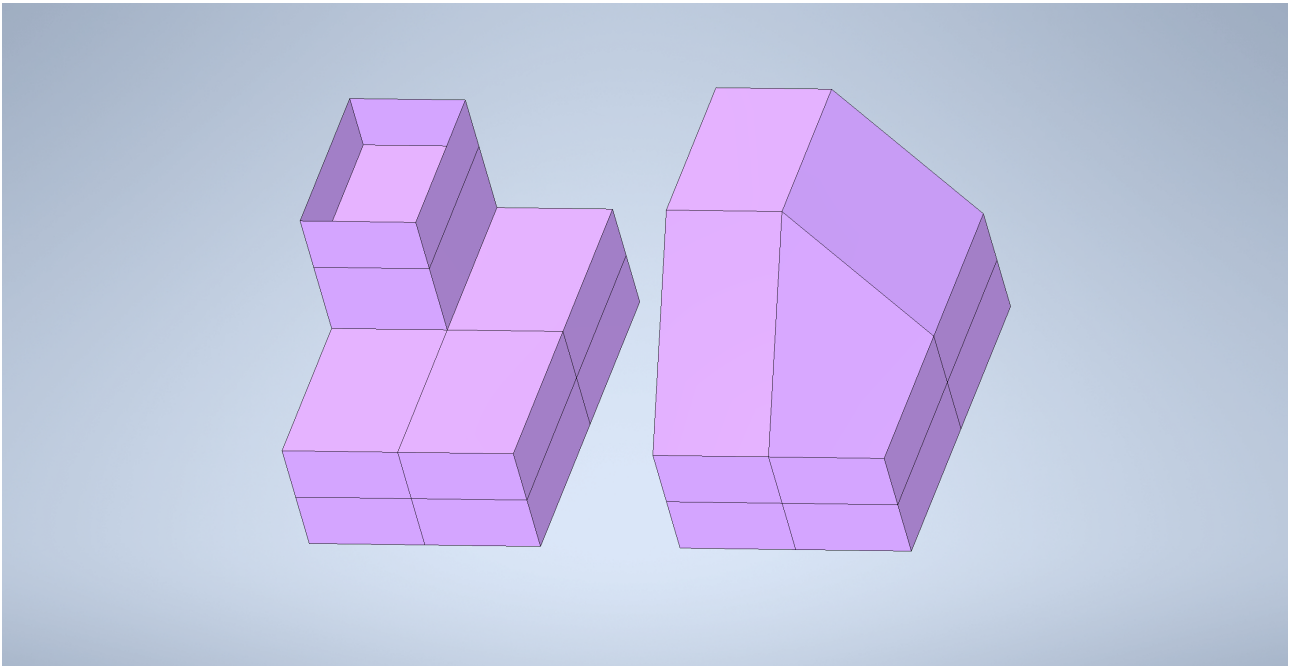
The Coordinate System for Transformations

An important note when editing freeform models is that the coordinate system used for transforming can be set to either the world coordinates, local coordinates or view coordinates. Most commonly the former two are used. We use world coordinates to move the geometry vetically or horizontally, while we use local coordinates to move orthogonally or tangent to the face - this is particulary useful for angled faces.

Soft Modification

Soft modifications allow for more gradual geometries, but as a trade off usually affect a wider amount of the freeform body. The type of modification (Circle, Rectangle, or Grow) determines the "range of influence" (i.e the area of the freeform body which is affected) and can be varied using parameters. For example, a $\boxed{\text{Circle}}$ modification with radius 10mm will affect any geometry within 10mm of the entity being edited. Usually, larger influence radii correspond to more gradual change. The $\boxed{\text{Falloff}}$ and $\boxed{\text{Gradient}}$ determine how the change will be distributed through this area.

Deleting Features

We delete features under $\boxed{\text{Freeform} \rightarrow \text{Edit} \rightarrow \text{Delete}}$. Typically, deleting edges will merge faces together, deleting faces will hollow out the freeform body and deleting points will merge all adjacent faces and edges.

Aligning Freeforms

We align points or symmetry planes to planes using $\boxed{\text{Freeform} \rightarrow \text{Edit} \rightarrow \text{Align Form}}$. Note that for freeform models, planes are more difficult to define because we can't create planes based on freeform geometry. Therefore, it is often practical to align freeforms to existing planes.

By doing this, we know where our freeform is in relation to the rest of the model, and can more easily create new planes (such as for symmetry operations).

Modifying Freeforms

This section covers the other tools used to modify freeforms.

Inserting Edges and Points and Subdividing

A common method used to modify freeforms is the introduction of additional geometry. This makes the the model more intricate and gives us greater control over how the blocky model translates to the smooth model. The three primary ways to introduce more detail is by inserting an edge, inserting points and subdividing faces.

In edge inserting (Modify → Insert Edge), we split a face by offsetting one of the edges that bounds it. The location (from 0 to 1) represents how far down the face to insert the new edge. We can also choose if we want the new edge on one or both of the faces the reference edge lies on.

If we want to make a new edge that is not parallel from an existing one, we can instead add two points and create a new edge between them (Modify → Insert Point) - we can also add a single point to bisect an existing edge.

To split faces into many parts, we use the (Modify → Subdivide) tool. This allows us to break down down a face into equidistant rows and columns.

For each of these tools, we can either choose the simple or exact mode. Simple Mode will create the exact number of faces specified, however in doing so may change the overall shape. If you only care about getting the number of faces you want, use this mode. If you want to refine the detail of an existing model without yet changing the overall shape, use Exact Mode. This adds extra faces to surrounding geometry in order to preserve the original shape.

Merging, Bridging and Creasing

Two sets of open edges (those connected to only one edge) can be joined together using the Modify → Merge Edge feature. This acts in a similar way to the Loft feature in 3D modelling, creating a bridge between the two edges. The merging point can be defined as being at the edge set or in the middle of them. This is particularly useful for joining two holes together - remember we can double click to select the entire loop of an edge.

Modify → Bridge perform similarly, however they join two faces together without having to create a hole first. Between two bodies, this will create a solid bridge, however within a single body this will create a hole between the two faces. We can bridge from any number of faces to any other number of faces, and customise the bridge by specifying the number of faces it will have and the number of twists it will perform. This is also very similar to the Loft feature. Note that by default the preview for this feature is disabled because it is computationally expensive. This can be changed in the dialog box.

Freeforms convert blocky models to smooth models, removing any sharp edges. However, we may sometimes want these sharp edges for particular purposes. The Modify → Crease Edges tool allows us to specify which edges should not be smoothened. They can be uncreased in like with Modify → Uncrease Edges .

Welding

If there are two or more points on the freeform that we want to connect, we can do so by using $\boxed{\text{Modifty} \rightarrow \text{Weld Vertices}}$. The weld can be made to be at the centre of the points or towards the second vertex. In addition, with multiple points the welds can be made to a specific tolerance.

An edge weld is defined as a loop which cuts the freeform object in two, and is often the result of either basic modelling or features such as mirroring. To break apart a freeform, we use the $\boxed{\text{Modify} \rightarrow \text{Unweld Edges}}$ and then select the loop we want to split along.

Flattening and Thickening

$\boxed{\text{Modify} \rightarrow \text{Flatten}}$ allows us to compress selected vertices into one plane. We can either define the plane we want the points to lie on, a corresponding parallel plane, or we can let Inventor compute a line of best fit automatically.

$\boxed{\text{Modify} \rightarrow \text{Thicken}}$ lets us add thickness to surfaces - in reality, this is not strictly true because the inside of our thicker surface is actually hollow. This tool takes a body and offsets it by a specified amount to add thickness (or remove existing thickness). This results in two copies of the same body, one slightly smaller inside the slightly larger one - this gives the thickness. We can choose not to join these together, or more typically join with either a sharp or a rounded edge. Typically this offset is applied in all directions (normal to the surface), but we can also choose a specific axis to add thickness to.
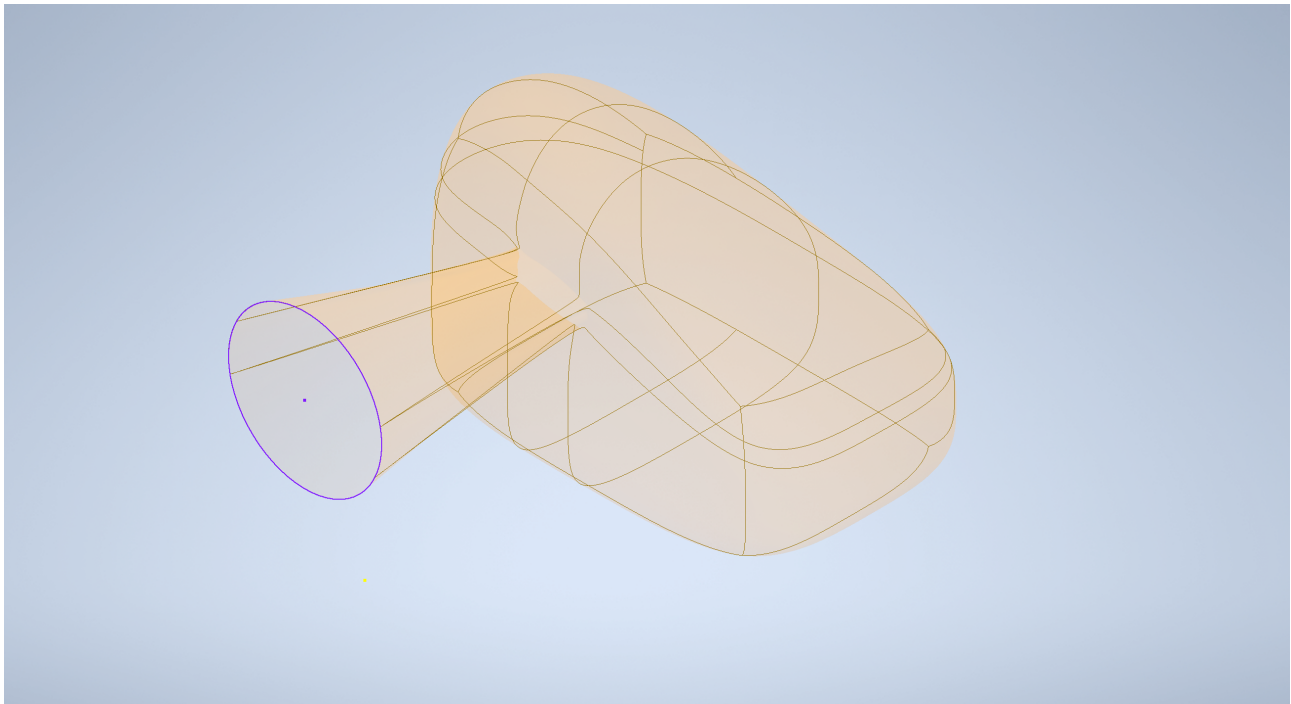
Matching

Edge Matching is a critical step in incorporating freeform models into CAD designs. Often, freeform models tend to be very quick to make, but difficult to do precisely - and are therefore difficult to design around. Usually what we do is design our freeform to what it approximately needs to be and then use this tool, $\boxed{\text{Modifty} \rightarrow \text{Match Edges}}$, to line the freeform up accurately. It glues an edge or loop from a freeform to a 2D sketch or 3D geometry, ensuring these models can be integrated into larger projects.

We can also set tolerances and flip the way to connect these edegs - for those familiar with topography, this creates a non-orientable surface homeomorphic to the mobius strip because we are introducing a twist. Therefore, we usually don't want to do this, and only use this setting when Inventor gets the default direction wrong. In addition, we usually avoid sharp corners because these are difficult for Inventor to model in 3D.

A special case arises if the edge of the freeform is from a NURBS (Non-Uniform Rational B-Spline) Surface. These surfaces are usually imported from other softwares or created in the Surfaces menu and then transformed into freeform. If we are using these types of surfaces, we can define the continuity of the matching to be G0, G1 or G2.

Note that matching edges will create a "Matches" folder under the freeform object in the Part Navigator, and if any of the sketches change the edge matchings can be rematched or redefined here. It doesn't do this automatically because of how computationally expensive this process is.

Symmetry and Mirroring

Like the corresponding tools for sketching and modelling, these tools allow us to define associative symmetries and mirrors in our freeform model. Under the `Symmetry` subpanel, we can `Symmetry → Mirror` a body along a mirror plane. Any changes to one side will be reflected in its mirror. Touching geometry across the mirror plane is typically welded together. The same effect of associative features can be achieved through the `Symmetry → Symmetry` tool. This adds symmetry to two already existing features. We can remove symmetries if needed as well with `Symmetry → Clear Symmetry` .

A further option is to make a freeform body uniform under `Check → Make Uniform` . This removes the pinch points and attempts to smooth out the surfaces as much as possible. Unless there is a good reason not to, we usually run this command at the end of our modelling to make sure everything is uniform.

How to Use Freeforms

Freeform modelling can be really good for certain parts of the design proces, but lacks the necessary precision for many engineering applications. To use them effectively, we recommend following the below process:

1. Identify where freeform modelling is useful. They are often used for complex 3D surfaces and shapes that don't necessarily need to be precise, or to join two components together. Freeform models also can't be dimensioned, so consider how a component is being manufactured.

2. Create the surrounding model that you want to integrate your freeform into. This should include sketches and faces for it to connect to. Also define planes and axes to guide your freeform modelling.

3. Choose the primitive that most closely aligns with that you want to design. Remember that for a spherical-based object that behaves the same in all directions, choose the quadball. Decide how detailed your model will be: the finer the mesh, the greater control you will have over the shape but the more computationally expensive it will be.

4. Define any symmetries or mirrors in the design. These should be used whenever possible to maintain symmetry of the design if needed.

5. Edit and modify your model as necessary.

6. Ground your freeform model to the correct location relative to your model using the Align Form tool.

7. Use the Match Edge tool to connect the freeform to the rest of the model precisely.

Glossary of Freeform Terms

Freeform object/model: a connected group of faces, edges and vertices/points.

Open surface: a surface which does not join in on itself to create volume. The boundary of this open surface is the edge that is only connected to a single face.

Loop: a series of edges connected by vertices which forms a closed chain (i.e forms the boundary of a surface)

Freeform geometry: the edges, points and faces that comprise a freeform model.

Mesh: the group of edges that make up the model.

Surfaces

Simplifying 3D Models

Plastic Parts

Insert

Harness

The Shape Generator

Shape Generator is Autodesk Inventor's version of generative design, which is a powerful feature which takes a load scenario and creates highly optimised structural designs.

Feature Management

Best Practices

Assemblies

Placing Components

Creating Components and Adaptivity

Relationships

Patterning Components

Productivity Optimisation

Simplifying

Contact Sets

Managing Components

Design Accelerators

Fasteners

Frames

Power Transmission

Springs

Inspecting and Analysing

Component and Assembly Stress Analysis

Autodesk Nastran

Graphical Analysis

Tolerance Analysis

Interference Analysis

Dynamic Simulation

Frame Analysis

Annotations

Annotations are tools that are used to add dimensions and other notes to a 3D model. In general, this approach is known as Model-Based Definition, and allows you to convey all the necessary information about a model without having to generate 2D sketches. Once you add annotations, they can be found in the Model Browser under the relevant folders.

Tolerances

When doing 3D modelling work, it is almost always necessary to incorporate tolerances to account for machining imprecisions.

General Annotations

General Dimensions

Add geometric dimensions to the 3D model by selecting $\boxed{\text{General Annotation} \rightarrow \text{Dimension}}$, in a similar manner to 2D sketches. After selecting the features to dimension, you can then customise the text (and add symbols) and modify the style type of the dimension. Dimensions can then also be dragged around after creation, and using $\boxed{\text{Right Click}}$ on an annotation

Holes and Threads

Dimension holes and threads by selecting $\boxed{\text{General Annotation} \rightarrow \text{Hole/Thread Note}}$. Pair this with the $\boxed{\text{3D Model} \rightarrow \text{Hole}}$ feature to ensure that all holes are properly documented.

Surface Textures

The $\boxed{\text{General Annotation} \rightarrow \text{Surface Texture}}$ feature creates annotations that describe how a surface is to be machined, textured and finished. The format of these annotations abides typically by the ISO standard.

Exporting Annotations

Inventor Studio

Creating Drawings

Presentations

Sheet Metal

chapterComputer Aided Manufacturing

BIM

Fusion Interoperability

3D Printing Compatability

Electromechanical Linking

Piping and Tubing

Cables and Harnesses

Mold Design

Weldments

Nesting

Task Scheduling, iLogic and Inventor SDK

Acknowledgements and Copyright