ΨΩM.A.R.S

Mechatronics and Robotics Society

MARS CAD Guide
Autodesk Inventor (R) 2023

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About This Guide

This is a guide developed by UQ MARS with the intention of broadening the awareness of the capabilities of CAD software. 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 those who have a minimal background in CAD and are able to utilise its basic functionality, but are not aware of the breadth of tools offered. This guide will aim to give a broad understanding of Autodek Inventor's various tools and in what applications they can be benefited from. It should be used as a reference guide to determine whether Autodesk Inventor has the required functionality and specific resources will be recommended for those who wish to use these tools. It should be noted that, although this guide is for Autodesk Inventor, most of the features presented will also be available in other software packages.

Using This Guide

Crash Course: Top 20 Takeaways

- Combine 2D profiles by using the Intersection Curve tool under 3D Sketch. This lets you create complex surfaces from only a side and top view of the object in question.

- Projecting geometry is a vital skill

- Utilising inferred constraints and snapping geometry to points will make your worflow exponentially faster.

- Use the Automatic Dimension tool!

- Parameters are very useful

- Know the process of downloading CAD files and creating components around them

- The Shape Generator can be very useful for advanced projects

- If you can, use Design Accelerators

- Don't forget that Inventor stores a wide array of physical properties

- Know how to suppress and hide features

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Work Features

2D Sketches

2D Sketching is one of the most fundamental parts of Autodesk Inventor, providing the baseline geometry for creating 3D features. They are used everywhere, from extrusion cross-sections to guide rails to surface boundaries.

Creating a Sketch

2D sketches can be created in any plane or flat surface, via any of the following ways:

- Clicking on a plane or flat surface and selecting "Create Sketch" on the popup ribbon

- Clicking on a plane or flat surface and selecting "3D Model ¿ Sketch ¿ Start 2D Sketch" or "Sketch ¿ Sketch ¿ Start 2D Sketch" on the window ribbon

- Right clicking on a plane or flat surface and selecting "New Sketch" on the popup wheel

- Selecting "3D Model ¿ Sketch ¿ Start 2D Sketch" or "Sketch ¿ Sketch ¿ Start 2D Sketch" on the window ribbon and then clicking on the desired plane or flat surface

- Right clicking on the model browser and selecting "New Sketch"

Adding Geometry

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.

- The bottom right of your workspace will display a lot of useful imformation 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.

- 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.

- Existing 3D objects can sometimes obstruct your view of your sketch. You can either change the camera angle or Right Click ¿ "Slice Graphics".

Types of Geometry

When selecting the start point of a line, you can click and drag on existing geometry to create a line which is tangent/perpendicular (see "Constraints"). 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 click 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. G2 Smooth Constraint).

- 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.

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 the cross-section that the sketch plane makes with any object. Can also be used to project the intersection of the sketch plane with a composite feature or surface.

- 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: —————————

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. Two parallel lines: the distance between them

2. Two nonparallel lines: the angle between them

3. Two curves/points: the distance between them. The movement of your cursor dictates whether this will be the vertical distance or horizontal distance. By clicking a third time in between the two points, the dimension will be the Euclidean distance.

4. 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 by selecting "Format ¿ Driven Dimension" or by right clicking and selecting "Driven Dimension".

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

Types of Constraints

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 inferred while placing geometry. For example, drawing a line which is horizontal will create an inferred horizontal constraint, and likewise for constraints such as parallel constraints. If you find that a constraint is being inferred with the wrong geometry (for example if you want to a parallel constraint to one line but it infers a parallel constraint with another), you can first mouseover ("scrub") the desired constraint 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 "Constraint Settings" under the "Constrain" tab. 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 "Automatic Dimensions and Constraints" under the "Constrain" tab, dimensions and constraints can be added to fully parameterise the sketch.

Formatting

The graphic display of geometry can be adjusted under the "Format" tab. 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 "Show Format" under the "Format" tab.

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.

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

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! Transform sketches into 3D features, surfaces and more.

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

- 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

Patterning 3D Features

Freeform 3D Modelling

Surfaces

Simplifying 3D Models

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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.

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