Autodesk Inventor Woordenlijst

In dit document vind u een verklarende woordenlijst van alle Inventor termen. Dit kan handig zijn om de reden van een foutmelding te begrijpen of commando's te leren kennen.

active part

The part that is being edited. In an open part file, the part is active and available for edit. In an assembly file, select the part in either the browser or the graphics window before it can be edited. If a part or subassembly was previously hidden or designated as background, it must be enabled before activation. The active part may be edited. See also background, enabled part, and undisplayed part.

active sketch

The current sketch to edit.

active standard style

In drawing documents, the default styles associated with the active standard.

adaptive compared to associative

Associative geometry has a one-way parametric relationship between parent and child geometry. Child geometry projected from a parent part is dependent on parent geometry. Child geometry updates when the parent is modified.

Adaptive geometry can have two-way dependencies. One part is designated as fixed geometry, and its adaptive counterparts update when the fixed geometry is changed. Whenever a part is reassigned from adaptive to fixed geometry, then changes made to that part update the remaining adaptive counterparts.

adaptive feature

A feature that can resize when constrained to other features. Individual part features may be designated as adaptive using the context menu. Features are constrained to a specified size, unless they are designated as adaptive.

adaptive part

Underconstrained part geometry can resize when designated as an adaptive part in an assembly. Assembly constraints position adaptive parts relative to other parts and adapt the part topology to fully constrained part features. Features that were underconstrained in the part file can resize according to assembly constraints and positions of other parts.

adaptive status

The status of a part that is not constrained to a specified size or shape. Parts are rigid bodies by default, but may be designated as adaptive in the context menu or the Occurrence tab of the Properties dialog box. Using assembly constraints to adjust size and shape relative to other parts may modify adaptive part topology. In assemblies, one occurrence sets the adaptive status for multiple placements of a component.

adaptive subassembly

In the context of an assembly, designation of a component that contains underconstrained parts or subassemblies. When an adaptive subassembly is constrained within its parent assembly, or to a component in another assembly, underconstrained geometry in an adaptive part resizes. For example, adaptive piston and rod subassemblies are sized and positioned when inserted in an air cylinder assembly.

adaptive work features

Construction geometry (such as work points, work planes, and work axes) within a part that is positioned by the geometry of other parts. For example, a 3D sweep feature relies on adaptive work points positioned relative to other components in an assembly to establish its position. The position of the work features adapts to changes made to the referenced geometry.

AEC Exchange Connector

A preparation tool that enables Autodesk Inventor content to be used in a connected system in mechanical, electrical, and plumbing (MEP) applications.

aligned dimension

A linear dimension parallel to a line spanning the minimum distance between two measurement points.

all components suppressed

Suppresses all children of the assembly, which results in a fast load of only the assembly file. Subassemblies are fully suppressed so that their browser structure is not visible. All Components Suppressed is the lowest-weight loading of an assembly.

all content center suppressed

Suppresses all components designated as content, and does not load their occurrences in memory. For example, consider the hundreds of fasteners in a design. If the Content Center files are stored in a remote location, reduced network traffic may also result.

all parts suppressed

Suppresses all parts at all levels of the assembly hierarchy. Subassemblies are loaded. You can investigate the assembly structure, constraints, and other attributes without loading part files.

all styles

Styles in the current document and the style library.

all-around symbol

An open circle at the vertex of the welding symbol that indicates the weld is to go around the entire joint.

alternate unit dimension

A dimension that displays two compatible units of measurement. Also called alternative dimension or dual dimension.

angle constraint

An assembly constraint that controls the angle between planes on two components in an assembly.

angle dimension

A dimension that denotes the angle formed by two lines.

arc

Geometry is defined four ways in sketches: center-point arc (specified by a center point and two points on a curve), three-point arc (specified by two endpoints and a radius), tangent arc (specified from the endpoint of a curve to another point), and an arc tangent or perpendicular to a curve (created by dragging with the line command).

array

See pattern feature.

arrow side

In weld situations such as a T-shaped joint, there are two potential places for a weld. The side of the joint to which the arrow points is known as the arrow side. The arrow side weld is made with the instructions given below the reference line for an ANSI symbol and above the reference line for all other standards.

aspect ratio

The proportional relationship of height and width. For example, a bitmap used in a sketch can be resized but it retains its original aspect ratio. The proportions of the height to width do not change.

assembly

Two or more components (parts or subassemblies) considered as a single model. An assembly typically includes multiple components positioned absolutely and relatively (as required) with constraints that define both size and position. Assembly components may include features defined in place in the assembly. Mass and material properties may be inherited from individual part files.

assembly constraints

Rules that determine how parts in an assembly are placed relative to other parts in the assembly. Constraints remove degrees of freedom. Assembly constraints include angle, flush, mate, and tangent. Constraints may be placed between faces of features, part edges, points, inferred axes, and part work features such as planes, axes, and points.

assembly features

Features (chamfers, extrude cuts, holes, weld beads) that are created and saved in the assembly file. The topology is not passed down to the component files. Used to define specific manufacturing processes such as match drilling and post-weld machining.

assembly pattern

In an assembly, components arranged in a circular or rectangular pattern. All elements in the pattern are identical. Assembly patterns are used to place multiple bolts in holes, or to position any component or components in a symmetrical arrangement. An assembly pattern can be associative to a feature pattern, updating when the feature pattern is edited.

assembly presentations

See presentation view.

assembly structure

The hierarchical tree shown in the browser that graphically illustrates relationships among components in the assembly model. The assembly structure shows the ownership of constraints and components within a subassembly. In general, component position in the assembly structure compares to the sequence of assembly during manufacturing.

associative design view representation

A drawing view that is associated to a design view representation in the assembly environment that updates automatically when changes are made to the design view representation in the assembly environment.

associative flat pattern views

In sheet metal design, flat pattern views are associated with the 3D model, like drawing views. When the model changes, the flat pattern view is updated. However, flat pattern dimensions do not update 3D models.

associative sketch pattern elements

In sketch patterns, all pattern elements are fully constrained as a group. Changes to an element update all other elements in the pattern. If association is removed during pattern creation or editing, constraints are removed. Pattern geometry becomes copied geometry with no association to other elements.

associative spline

When creating an offset from an ellipse, an associative spline is created instead of a true mathematical ellipse if the offset selection point does not display the parent ellipse axis. The result is a spline equidistant from the ellipse.

asymmetric

An option in both the Extrude and Revolve commands that allows a 2D profile to be extruded or revolved in both positive and negative directions simultaneously with different linear or angular values.

attribute

A data management description of information associated with a part of subassembly. In an assembly or a bill of material, an attribute typically refers to a file-level property to incorporate into a structured bill of material.

AutoCAD spline fit method

Creates an Autodesk Inventor spline using the same method used by AutoCAD.

automatic blending

A re-blending technology that automatically moves adjacent tangential faces and also creates new blends if required. It is active by default in the part modeling, assembly, and solid edit environments for the Move Face and Thicken commands.

automatic edge chain

A selection setting used during creation of 3D fillets. When Automatic Edge Chain is on, selecting an edge automatically selects all edges tangent to it.

axis of revolution

The centerline of a revolved feature.

axonometric projection

A type of orthographic view created by projection lines that are perpendicular to the viewing surface or plane, where a 3D model appears as inclined, exposing the X, Y, and Z view planes simultaneously. For example, an isometric view is axonometric.

base feature

The first feature created in a part. May be an imported base solid (.sat or .step file format), in which case the base feature is a fixed size. Sketched or placed features add details to the base feature and are positioned relative to one another using dimensional or geometric constraints. The base feature should represent the most basic shape in the part.

base solid

A model in SAT or STEP file format imported in Autodesk Inventor as a base feature (the first feature in a model). A base solid is fully dimensioned; its size can be changed only by commands in the Solids environment, but the solid can be modified by adding features.

beam

Typically a horizontal structural member of a frame often analyzed for load bearing capacity.

bend

In sheet metal design, a corner seam can be converted to a bend. A bend is the small radius between faces that represents the fabrication process in which bends are applied to a flat pattern to form a 3D part.

bend allowance

In sheet metal design, a factor applied when converting a bend to an unfolded condition. Material thickness, material type, and bend radius are used to calculate the amount of distortion.

bend lines

In sheet metal design, bend lines can be displayed in the flat pattern drawing view to show the start, center, and end of the bend zone.

bend relief

In sheet metal design, a small notch cut at the end of a bend to prevent material from tearing.

bend zone

In sheet metal design, the area where material deformation takes place, often referred to in the flat pattern as the area between bend lines.

bill of material

For an assembly, a document that describes components details such as component name, material, and quantity. Bills of material can be single or multiple levels (exploded).

bottom-up design

A sequential assembly design technique that adds previously designed components to an assembly model and positions the components using assembly constraints.

bounding rectangle

A virtual rectangle that represents the extents of an image in Autodesk Inventor. When an image is placed in a sketch, constraints and dimensions can be applied to the bounding rectangle to position the image.

broken view

A drawing view that was divided with cosmetic break lines and foreshortened. It allows a component view and its critical annotation areas, which in an unbroken view are prohibitively small, to display at a legible scale.

browser

The graphical hierarchy showing relationships among geometric elements in parts, assemblies, and drawings. Icons represent sketches, features, constraints, or attributes for each model. Objects are shown in the browser in the order in which they were created. Objects can also be edited, renamed, added, deleted, copied, and moved to a different location in the browser.

cached data

In the computer memory, the last used solution that yielded usable sketch or system data. For example, when a part containing an iSketch is edited outside of the parent assembly, the assembly displays a message warning that changes may not be reflected in the part because the last-known (cached) data is used to represent the part in the assembly.

catalog

Microsoft Windows folder hierarchy. Catalog folders are used to manage iFeatures in a user-defined hierarchy.

center point

A work point placed at the intersection of the default reference planes in a part or assembly file.

centerline

The axis about which a profile is revolved (revolve feature) or swept (sweep feature). To automate centerline selection, right-click a curve, and then change the linetype to a centerline.

chamfer

A placed feature that bevels a part edge and is defined by its placement, size, and angle.

child

In a hierarchical design relationship, a child element is dependent on another (parent) element. A typical example is a feature, such as a cut, that depends on a base feature. In the browser, the child, or dependent feature, is indented under its parent. A child feature can be a parent to other features. In most cases, deleting the child feature has no effect on the parent feature.

circle

Geometry defined two ways in sketches: a center-point circle (defined by a center point and a point on its circumference) and a tangent circle (defined as tangent to three lines on its circumference).

coincident constraint

A geometric constraint that constrains two points together or constrains one point to a curve. When this constraint is applied to the center points of two circles, arcs, or ellipses, the result is the same as the concentric constraint.

collinear constraint

A geometric constraint that causes two or more line segments or ellipse axes to lie along the same line. In an assembly, a collinear constraint is achieved with a mate constraint between two lines, edges, or axes.

combine

Performs a cut, join, or intersect operation on selected solid bodies.

component

A part or subassembly placed into another assembly. Assembly components can be single parts or combined parts that operate as a unit (or subassembly). Components can be treated as parts within other assemblies.

composite iMate

A group of individual iMates, selected and named in the browser, that are to be used as a unit. In an assembly, when a component with a composite iMate is matched with another component with a composite iMate of the same name, all constraints in the iMate collection solve at once.

concentric constraint

A geometric constraint that causes two circles, arcs, or ellipses to have the same center point. The result is the same as that of a coincident constraint applied to the centers of the curves.

configuration file

A file that specifies the options for opening AutoCAD or Mechanical Desktop files, or saving files in DWG or DXF format.

connection point

A point that you can define for use in inferencing or snapping to other geometry. You can add connection points and use them to position sketched symbols, title blocks, and other drawing objects.

constant-radius fillet

A fillet that has the same radius along its entire length. Use the Constant tab on the Fillet feature dialog box to select the edges and specify the radius for a constant-radius fillet feature.

constraints

Rules that govern the position, slope, tangency, dimensions, and relationships among sketch geometry or the relative position between parts in an assembly. Geometric constraints control the shapes and relationships among sketch elements or assembly components. Dimensional constraints control size. Applying constraints removes degrees of freedom.

construction environment

Imported data resides in the construction environment until it is promoted to the part environment. For example, when importing data in IGES or SAT format, a Construction icon is placed in the browser, with data groups nested below it. In the active construction environment, commands modify the geometry, such as repairing errors in imported data.

construction geometry

Geometry used to assist in the creation of sketches or features, but not used to define profiles or paths. The line style designates a curve as construction geometry.

consumed sketch

A sketch incorporated into a feature, such as a sketch used in an extrusion. By definition, the sketch is consumed by the feature.

containing selection window

Dragging your cursor from right to left creates a crossing selection. Objects that are fully within the containing window are selected.

context menu

The menu that appears when the right mouse button is clicked. Displays commands and Help for the current operation.

coordinate system

The set of magnitudes (visually represented by the X, Y, and Z axes) that determine the position of points, lines, curves, and planes in part and assembly files. By default, a grid displays on the active sketch plane of the coordinate system.

corner break

In sheet metal design, a corner break applies either a radius or chamfer edge to the end of a flange.

corner relief

In sheet metal design, a small notch that prevents material from tearing is placed where two bends join.

corner seam

In sheet metal design, a corner seam mates edges where two bends join, including the alignment or overlap of edges, miter of angled flanges, and inserts corner relief.

cosmetic welds

In a weldment model, cosmetic welds are graphic elements that indicate welded edges. Cosmetic welds represent a wide range of weld types. They do not add mass to the assembly and cannot be used in interference checking. Cosmetic welds always reside in the Welds feature group.

curves

Geometric objects in a sketch, including lines, arcs, circles, splines, and ellipses.

custom iPart

An iPart factory generates iParts, each of which has a unique instance with parameters, properties, and other values that are maintained in an embedded spreadsheet. Each row in the spreadsheet represents an individual iPart with a unique instance. When a custom iPart is used in an assembly, specific values specified during the iPart definition can be modified (such as length, width, or thickness).

cut

In sheet metal design, cuts are sketched shapes that define material to remove from a single face, a specified depth, or an entire 3D part.

cut length

For frames, the overall length of the raw stock required to create a structural member (including enough material to accommodate end treatments on either end).

cut list

A cut list is a table listing components with dimensions and cut treatment information for fabrication and accounting purposes. Each treatment on a frame member produces a custom property called CUTDETAIL(n), which you can add as manufacturing notes in the cut list.

cut operation

One of 3 Boolean operations (cut, join, and intersect) that define the relationship between a sketched feature and an existing feature. A cut operation removes the volume of a sketched feature from an existing feature. Not available for base features.

cutting plane

Used to display an assembly section temporarily for defining new features. A cutting plane is an existing plane defined by default work planes within the assembly or within one of the component parts of the assembly.

deck

The walking surface of a platform, typically constructed of diamond plate or expanded metal.

degrees of freedom

The variables by which an object can move. In assemblies, a body floating free in space with no constraints to another grounded body can be moved along three axes of translation and around three axes of rotation. Such a body is said to have six degrees of freedom. Constraints remove degrees of freedom by restricting the ways sketch geometry can change or a free-floating body can move.

demote

In the browser hierarchy, dragging a feature or component farther from the top level.

dependent features

Features that are geometrically dependent on another feature, such as chamfers and fillets.

derived part

A new part that uses an existing Autodesk Inventor part as its base feature. A part can be scaled or mirrored when inserted into a file, and additional features can further modify the body. A derived part is linked to the original part and can be updated to reflect changes to the original part.

derived part feature

A base feature in a new part that is derived from an existing Autodesk Inventor part. When creating a derived part, you import a part as the base feature. The base feature is listed as a derived part feature in the browser. Part modeling features can further modify the derived part feature.

design elements

See iFeatures.

design layout

A 2D master drawing or sketch that is drawn to scale with minimal detail to capture the known design parameters of a problem or project. The design layout is a dynamic document that becomes less useful as a project nears completion. It is superseded by detailed part and assembly designs.

design properties

The file properties that can be used to find and manage Autodesk Inventor files. Design properties are also used to maintain and update information automatically in title blocks, parts lists, bills of material, and sketched symbols. See also iProperties.

design view representation

A named and saved assembly view where components have characteristics selectively designated, usually to present an uncluttered working environment. A design view captures characteristics such as selection status (enabled or not enabled), visibility status, color and style, zoom magnification, and viewing angle. The design view representation can be named, and when recalled, components are shown with the designated display characteristics.

diameter dimension

A dimension that controls the size of a circle or arc in a sketch or drawing view. Can be stated as a numeric constant, as a variable in an equation, or in a link to a parameter file. In drawing views, can be designated as a driving dimension (resizes the model) or reference dimension (does not resize the model).

dimension input

When Dynamic Input (Heads-Up Display) is enabled, the value input fields display polar coordinate (distance and angle) values when a command prompts for a second point. The dimensional values change as you move the cursor. Press TAB to move to the value you want to change.

dimension style

A group of settings that control the default format for dimensional annotations in drawings. You set up dimension styles in a template or drawing before you place dimensions, hole notes, and other annotations.

dimensional constraints

Parametric dimensions that control sketch size. When dimensions are changed, the sketch resizes. Dimensional constraints can be expressed as numeric constants, as variables in equations, or in parameter files.

Direct Manipulation

An interface that allows the user to interact and modify a model via a manipulator, while viewing the changes in real time. The resulting interaction is dynamic, visual and predictable. The user focuses on the geometry in an in-canvas display instead of interacting with user interface elements such as the ribbon, browser, and a dialog box.

See in-canvas display, manipulators, mini-toolbar, selection tags and value input box

downstream process

An operation that follows the selected operation in an order-dependent process. For example, post-weld machining operations are downstream from the weld bead and weld preparation operations.

draft

See face draft.

draft angle

A taper applied to part faces, for example, to allow extraction from a mold. Draft angle can be applied when a sketch is extruded or can be applied to selected faces.

draft plane

A planar face used to determine the direction of the draft angle.

draft view

A special view in a drawing that does not contain a representation of a 3D model. A draft view has one or more associated sketches. Place a draft view and construct a drawing without an associated model, or use a draft view to provide detail that is missing in a model.

When you import an AutoCAD file to an Autodesk Inventor drawing, the data is placed in a draft view. Dimensions, text, and other annotations are placed on the drawing sheet and geometry is placed in the associated sketch

drafting standards

Specific guidelines for drawing views, dimensions, notes, and annotations that ensure uniform presentation of information. Drafting standards facilitate the exchange of information among several users, vendors, and manufacturers.

drawing resources

The sheet formats, title block formats, border templates, and sketched symbols defined in a drawing or drawing template. Design resources are listed in the Drawing Resources folder in the browser. You can define new drawing resources and add them to the folder.

Use drawing resources to add new sheets, title blocks, or borders to a drawing, or to add custom annotations to a drawing sheet or drawing view.

driven dimension

A nonparametric dimension enclosed in parentheses that shows the current value of geometry. Its value updates when the geometry changes size, but it does not resize geometry.

driving dimension

A parametric dimension that determines the size of sketch geometry and resizes the sketch when its value changes.

duplicated feature

A feature that has been copied and arrayed in a rectangular or circular pattern or mirrored.

Dynamic Input

See Heads-Up Display (HUD)

edge

The curve joining two model faces.

edge chain

In a fillet feature, chains together all edges that are tangent to the edges you select to fillet.

edge set

A group of edges selected during the creation of a 3D fillet feature. The fillet radius and all fillet options are the same for all edges in a single edge set.

edit in place

A component is activated (the edit target) and edited in the assembly context. Except for the active component, all components are shown in wireframe and cannot be selected. Edits are saved to the part or subassembly file. Double-click the top-level assembly in the browser to reactivate the assembly.

edit target

The file to which edits are saved when constraints are applied or when assembly components are created or edited in place. The edit target can be a part file, subassembly, or top-level assembly that is activated in place.

ellipse

Geometry created from the intersection of a cylinder and an angled plane.

enabled part

In an open assembly file, a part that is fully loaded in memory and can be activated in place for editing. Placed parts are enabled automatically.

end treatment

The geometry of a structural member where it joins another structural member of a frame. End treatments can include miters, coping, butt, and others.

endpoint

An implicit point on the ends of curves. Endpoints can be included in constraints, dimensions, and as snap points.

engineering notes

Notes attached to a feature, part, assembly, assembly constraint, or other object to document the design process and decisions. Using Autodesk Inventor, you can capture, edit, and manage engineering notes.

equal constraint

A geometric constraint that causes selected arcs and circles to have the same radius or selected lines to have the same length.

equations

Mathematical expressions that range from simple numbers to complex algebraic and trigonometric expressions that use parameters as variables. Each parameter has a unique equation. You can enter an equation as the value of a dimension, feature, offset, or parameter.

Autodesk Inventor equations support most common mathematical operators but do not support addition or subtraction in exponents. Use the expression ul/units to create a negative exponent (for example, ul/min equals min-1).

exploded view

A view in which parts of an assembly are separated for unobstructed viewing. The direction and distance that parts can be moved are dependent on the settings that you specify when setting up the view. Exploded views are defined in an assembly presentation file and then used to add exploded views to a drawing.

expression

A mathematical expression relates one set of terms to a syntactical collection of other terms through mathematical operations, functions, and Boolean logical operators. Unlike an equation (whose sides are equal), an expression can include inequalities of greater than, greater than or equal to, less than, less than or equal to, and other relationships.

external frame

Frames often required for platforms, access ways, and stairwells used to operate and maintain complex machinery. You can build external frames either at the time machines or equipment are installed, or concurrently with the machinery.

extract

The process of copying a feature out of a part model and storing it in a feature file. An extracted feature can be retrieved from a Windows folder (catalog) and placed in a part file.

extruded feature

A feature created by adding depth to a sketched profile. Feature shape is controlled by profile shape, extrusion extent, and taper angle. Unless the extruded feature is the first feature, its relationship to an existing feature or body is defined by selecting a Boolean operation (join, cut, or intersect) and the participating bodies for the operation if multiple bodies exist. Optionally, can create a new body.

face

A bounded surface that defines part of an object, such as a part face or a face in a quilt.

face draft

A taper applied to part faces to cant a surface or allow a part to be extracted from a mold. Draft is defined by specified faces, pull direction, angle, and fixed edge or tangent surface.

feature

Parametric geometry that creates or modifies parts or assemblies. Relationships among features are defined by geometric and dimensional constraints. Types include sketched, placed, and duplicated features, work (construction) features, and assembly features. Features combine to build up a complex part or assembly model. Individual features can be modified as needed.

feature constraints

See constraints.

feature file

A file created when you extract a feature, sketch, or subassembly to be used as an iFeature. The iFeature is stored in a feature file (.ide) and can be retrieved by name and placed in a part or assembly file.

feature participant

See participant component.

feature sketch edit

The Edit Sketch command modifies a feature by changing its sketch dimensions. When Edit Sketch is selected in either the graphics window or the browser, the feature (and all features created after it) is temporarily hidden and its original sketch appears. After the sketch is modified, the feature updates to incorporate the changes and all subsequent features reappear.

fillet curve

An arc of a specified radius placed at the corner or intersection of 2 lines using the Fillet sketch command.

fillet feature

A placed feature applied to edges and corners of a 3D model. A fillet feature is defined by its type, radius, and placement.

fillet weld

See solid welds and cosmetic welds.

first-level component

A component linked directly to the top-level assembly icon in the assembly browser. A first-level component can be an individual part or a subassembly that contains multiple parts or other subassemblies.

fit point

A definition point that does not move when a spline is solved. Beginning and endpoints, and internal points constrained to other geometry, are fit points.

flag

A symbol extending out of the vertex of the welding symbol which indicates that to make the weld onsite. Welding symbols with no flag indicate to apply the weld in the shop.

flexible assembly

A subassembly that can be positioned in a top-level assembly while allowing its underconstrained positions to be driven or dragged from its parent level. The capability is useful for kinematic mechanism design.

flush constraint

An assembly constraint that points the surface normals of selected faces in the same direction. Placing a flush constraint removes three degrees of freedom (two rotational and two translational). In the Mechanical Desktop, a flush constraint is achieved using an assembly mate constraint and selecting a surface normal solution.

fly through

A motion-oriented view of a part or assembly model. In Perspective Camera mode, you can use the mouse and key commands to navigate internal portions of a model with full range of motion between X, Y, and Z axes.

foreign component

Parts and assembly models created in another CAD system.

frame

A mechanical component typically composed of welded structural members which serves as the foundation component of a machine or as an external access platform. Frames are typically a single item within a bill of material.

frame skeleton

A 2D/3D wireframe or solid body representation of the frame that consists of straight line segments.

Gaussian curvature

The Gaussian curvature of a surface at a point is the product of the principal curvatures at that point. The tangent plane of any point with positive Gaussian curvature touches the surface at a single point, whereas the tangent plane of any point with negative Gaussian curvature cuts the surface. Any point with zero mean curvature has negative or zero Gaussian curvature.

geometric constraints

Rules that define the geometric relationships of sketch elements and control how a sketch can change shape or size. Some constraints are inferred according to sketched shape and others may be manually applied to remove degrees of freedom. Geometric constraints are coincident, collinear, concentric, equal, fix, horizontal, parallel, perpendicular, tangent, and vertical.

global parameters

Called global variables in the Mechanical Desktop.

graphics window

The active modeling area in which sketches, constraints, features, parts, and assemblies are created and edited. In the graphics window, models can be rotated, zoomed in and out, and view characteristics such as color, material, and light defined.

ground shadow

Shadow cast below a visible model is updated dynamically as the model rotates or moves. The effect is similar to a shadow cast on the ground beneath a model.

grounded component

A part or subassembly for which all six degrees of freedom were removed relative to the assembly origin. You can position the part or subassembly without reference to other parts. It is fixed in space. The first part or subassembly placed in an assembly file is grounded automatically, although the ground can later be deleted and relocated, if needed.

grounded work point

A work point created in a part or assembly to represent a point fixed in space. A grounded work point is indicated in the browser by a push pin symbol. In a part file, the 3D Move/Rotate tool displays when point is created and provides specified transforms from the fixed point. 3D Move/Rotate tool is not available in assemblies.

Heads-Up Display (HUD)

Also referred to as Dynamic Input, the Heads-Up Display (HUD) in the Sketch environment is a user interface near the cursor to help you keep your focus in the sketching area. Value input fields near the cursor display information that is dynamically updated as the cursor moves.

When a Line, Circle, Arc, Rectangle, or Point sketch command is active, the value input fields provide a place for user entry. You can toggle back and forth between the value input fields by pressing the TAB key.

heal

Autodesk Inventor can extend adjacent faces until they intersect, or heal, when a face such as a fillet face is deleted. Healing is not possible if, for example, a cube face is deleted.

hidden folder

A folder displayed in the browser for each sequence in an assembly presentation. You can drag components into the hidden folder for a sequence to hide them for the duration of that sequence in the animation.

hole

A geometric feature defined by hole type, placement, size, and dimensions. A hole requires a center point sketch for its origin and placement.

hole tag

An alphanumeric identifier added to each hole included in a hole table.

horizontal constraint

A geometric constraint that positions selected lines, ellipse axes, or pairs of points parallel to the X axis of the sketch coordinate system or positions 2 selected points (curve endpoint, center, midpoint, or sketch point) the same distance from the primary axis (same Y coordinate).

horizontal dimension

A dimension that controls the size of curves that are parallel to the X-axis in sketches and drawing views. Can be stated as a numeric constant, as a variable in an equation, or in a link to a parameter file. In drawing views, can be designated as a driving dimension (resizes the model) or a reference dimension (does not resize the model).

iFeatures

Features, sketches, or subassemblies that can be used in more than one design are designated as iFeatures and saved in a file with an .ide extension. To add an iFeature to a part, use Windows Explorer to drag the file name and drop it in the active part file. To change the size of an iFeature, edit its sketch or feature definition or link it to parameters that define its size. You can precisely position an iFeature using geometric constraints and dimensions.

iMate

One half of a constraint defined in a component file. Used to position the component automatically in an assembly when matched to an iMate with the same name on another part. Can be given a unique name to describe its use.

iMate definition

Created when you create or infer iMates on geometry. When matched in an assembly, the solution is an iMate result.

iMate result

Assembly constraints that consume one or more iMate definitions.

in-canvas display

A user interface that is overlaid on the viewport to support Direct Manipulation operations. The in-canvas display typically features a mini-toolbar featuring command options, manipulators, and selection tags to allow direct and predictable interaction with the 3D model.

in-line work feature

A work feature created during the creation of another work feature. For example, when creating a work point, right-click and choose to create either a work plane or a work axis. You can continue to select and create in-line work features until the requirements for the work point, in this case, are satisfied. In-line work features are dependent on the active work feature command in which they were created.

in-place activation

The context switch from a parent-level assembly to a child of that assembly, such as a subassembly or an individual part. The activated subassembly or part may be edited, moved, constrained, or otherwise modified. Activating a part, subassembly, or assembly in place changes the edit target to the active object.

included geometry

Model edges, vertices, 2D lines and arcs inserted from existing parts into a 3D sketch using the Include command. Included geometry is converted to 3D geometry and can be used in a path sketch for a 3D sweep feature.

included path file

A secondary project file specified in a project. When an included path file is used, Autodesk Inventor searches the locations specified in both projects to resolve links and references between files. An included path file is often used to specify work group locations on a network.

inconsistent constraint

A constraint that conflicts with a constraint already placed and, thus, cannot be solved. In assemblies, examples of inconsistent constraints include a requested mate constraint when a flush constraint is required (inverted face normals), a rigid body topology conflict, or the adaptive status not set.

infer iMates

A process by which assembly constraints are converted to iMate definitions stored with the part.

insert point

A user-defined point at which the cursor is attached when a sketched symbol is inserted into a drawing. If you do not specify an insert point for a sketched symbol, the cursor is attached to the center of the symbol geometry.

interference

The volume created when portions of multiple components occupy a common space. The volume shared by the intersecting components displays; individual components can be modified to eliminate interference.

intermittent weld

A weld that is not continuous along its length. Two or three values are required depending on the standard. Possible values are length, pitch or spacing, and number.

internal frame

A frame you can use as the base upon which to build a machine. You can weld frames together, machine and then paint or otherwise finish them, or bolt them together using materials left unfinished, such as stainless steel or aluminum.

intersect operation

One of 3 Boolean operations (cut, join, and intersect) that define the relationship between a sketched feature and an existing feature. An intersect operation creates a feature from the shared volume of a sketched feature and an existing feature. Material not included in the shared volume is deleted. Not available for base features.

invisible part

See undisplayed part.

iPart

A part generated from an iPart Factory, whose multiple configurations are each maintained in a row in an embedded spreadsheet defined in the iPart Factory. The designer of the iPart Factory specifies parameters, properties, features, iMates, and other values to include or exclude from individual iParts. Standard iParts cannot be modified; Custom iParts contain some values that may be modified when the part is used.

iProperties

File properties that can be used to find and manage Autodesk Inventor files. iProperties are also used to maintain and update information automatically in title blocks, parts lists, bills of material, and sketched symbols. See also design properties.

isolated environment

A collaborative work environment in which designers copy all files from the server into a personal workspace before working on them. The project file for an isolated environment specifies the personal workspace and any additional local paths.

isometric

A type of orthographic projection where a 3D model is situated so that each of the three fundamental model planes (X, Y, and Z) has an equal degree of exposure relative to the view angle of the observer.

joggle

In sheet metal design, a means to offset material so that material can be overlapped.

join operation

One of three Boolean operations (cut, join, and intersect) that define the relationship between a sketched feature and an existing feature. A join operation adds the volume of a sketched feature to the existing feature. Not available for base features.

joint

The location where one structural member becomes coincident with a second structural member. In many frames, these locations define welded conditions. Joints are most common at either end of a structural member.

key values

A value used to define an iPart or a table-driven iFeature instance when used in a model. An iPart or table-driven iFeature must have at least one primary key and as many as eight secondary keys. Keys are numbered when defined in the iPart factory and control available values when the iPart is used. For example, selecting a primary key value filters available secondary key values.

keyframe animation

An animation created by arranging objects and taking snapshots of them at key moments during a sequence of movement or change. These key moments or keyframes become the fixed points in time through which the animation passes.

ladder

Platform frames often exist on multiple levels surrounding large equipment, and these levels often require ladder access. Ladders can consist of several parallel vertical structural members connected with various types of rungs.

leaf part

A single component in the assembly tree or a component in an assembly.

library feature

See iFeatures.

library search paths

The locations of files not edited. Libraries can include purchased or standard parts, Mechanical Desktop parts used in Autodesk Inventor assemblies, iPart factories and members, or other internally developed standard parts. A library is often referenced by multiple projects. Each project specifies the locations of its libraries.

line

Straight curve bounded by two endpoints. The line command on the Sketch tab chains line segments together and creates arcs tangent or perpendicular to existing curves. Segments and arcs are automatically joined by coincident constraints at their endpoints.

local style

A style cached in the current document. By default, the styles associated with a selected drafting standard are available in a document.

lock icon

Lock icons in the Dynamic Input (Heads-Up Display) value input fields constrain the cursor by the values that you enter.

loft feature

A sketched feature specified by tangency condition, termination, and order of sketches. Loft features blend 2 or more dissimilar sketch shapes on nonintersecting sketch planes. Optionally, can create a new solid body.

loop

A closed sketch shape that can include sketched curves, edges, and planar faces as boundaries.

lump

A group of faces that may contain some or no void. A part or surface body should have at least one lump. For example, if you draw two separate rectangles in a sketch and extrude both of them, the resulting part has two lumps. If you create two separate lines (not connected) and extrude them, the resulting surface has two lumps.

machining features

Assembly features added to the Machining folder in a weldment assembly. Machining features are added after weld beads are applied to the weldment assembly, often affecting multiple assembly components. Holes and extrude cuts are typical post-weld machining features.

machining features group

A folder in the weldment environment representing metal removal operations woerformed after welding. Machining operations often affect multiple assembly components. Holes and extrude cuts are typical post-weld machining features.

manipulators

In-canvas interactive objects that allow the user to easily manipulate objects for various modeling and editing tasks. Some examples include:

1) a distance arrow to dynamically drag the extrusion distance of one or more 2D profiles.

2) a rotational arrow to dynamically revolve 2D profiles around a single axis.

3) a sphere to locate the center of a hole, or to adjust the taper angle of an extrusion.

mask

A property that describes the transparency of an image inserted in a sketch. When a mask is applied to an image, the mask color is determined by the bottom right pixel of the image.

mass properties

See physical properties.

mate constraint

An assembly constraint that joins elements together with a surface normal orientation and an optional offset. A planar mate constraint usually moves two external part faces so that their surface normals point in opposite directions. Mate constraints can be used to join points, lines, edges, or axes together and to adapt diameters of unconstrained cylinders.

material properties

Properties defined by the material definition stored in individual part files. Properties include Young's modulus, Poisson's ratio, specific heat, density, yield strength, ultimate strength, linear expansion, and thermal conductivity.

Mechanical Desktop part

Component models created using the Mechanical Desktop.

middle-out design

An assembly design technique that combines aspects of both top-down and bottom-up techniques. Assemblies designed with the middle-out technique usually have some fixed equipment or requirements, but also require custom design solutions.

midpoint

The center of a line segment.

mini-toolbar

Located in close proximity to a selected object in the graphics window, the mini-toolbar displays in-canvas buttons that allow quick access to frequently used commands and command options.

minimum energy spline fit method

Generates a smooth curve with high visual appeal and better curvature distribution. Because splines contain more data, surfaces generated from the minimum energy splines compute slower than other spline curvature types.

missing geometry

A constraint error that occurs in an assembly when a component is constrained in position, but subsequent modifications are made to one of the constrained parts. The geometry involved in the constraint is no longer available. It can happen if the geometry was consumed by a subsequent feature operation or if the feature containing the geometry was deleted or suppressed.

missing tooth

In features or components arranged in a pattern, a position within the logical definition that denotes a skipped or suppressed occurrence.

motion constraint

Specifies intended motion ratios between assembly components, either by rotation (such as gears and pulleys) or rotation and translation (such as a rack and pinion). The first selected component moves relative to the second selected component.

multi-body part

A part file that contains more than one solid body. Each body can contain an independent collection of features or shared features. Each solid can be exported as a separate part file at the end of the design process.

multiview projection

A set of single plane orthographic views used to display an object through one view plane per projection. For example, a first angle projection is one view in a multiview projection set.

native component

A part or assembly created using Autodesk Inventor. A native component can be created in a part file or in place in an assembly file.

nested profile

A profile enclosed within the boundary of another profile. Nested profiles can be selected and extruded in a single command operation. For example, concentric circles can be selected and extruded to form a tube.

new solid

An operation in a part file that creates a new body. The solid body can contain a unique set of features separate from other bodies in a multi-body part file. Features can also be shared with other bodies.

not enabled part

A part not required for editing in the current design session but that is provided for context. Often, parts that are not enabled are completely positioned and were placed early in the assembly design process. In an open assembly file, parts are designated as not enabled in both the graphics window and the browser and may not be selected.

numeric constant

A quantity taken to have a fixed value in a specific mathematical context. Examples of numeric constants include pi, English and metric conversion factors, and the freezing point of water.

occurrence

Placement of a component, usually in multiple-level assemblies, where a component is replicated as a result of multiple placements of a single part or subassembly.

offset geometry

Duplicates and positions selected sketch geometry a specific distance from the original. By default, offset geometry is constrained equidistant from the original geometry.

OLE object

A linked or embedded object that originates in an application external to Autodesk Inventor.

origin indicator

A marker in a drawing view that sets the origin datum point for ordinate dimensions or a hole table. The origin indicator for a view is placed when the first ordinate dimension or hole table is added. A view can have only one origin indicator.

orthographic camera mode

A display mode where all points of a model are projected along parallel lines to the screen.

other side

In weld situations such as a T-shaped joint, there are 2 potential places for a weld. The side of the joint opposite the side to which the arrow points is known as the other side. The other side weld is made with the instructions given above the reference line for an ANSI symbol and below the reference line for all other standards.

pad

Typically a piece of flat stock added to the end of a structural member to provide stability where the member is bolted to another. Often, pads are added along the length of a structural member to provide additional material to be machined to close tolerances where additional components of an assembly are attached.

parallel constraint

A geometric constraint that causes two or more lines or ellipse axes to have the same slope and orientation.

parallel dimension

Linear dimension that shows the distance between two lines and has the same slope as the lines.

parameters

Used to define the size and shape of features and to control the relative positioning of components within assemblies. Can be expressed as equations to define the relationships between geometric elements relative to one another. Changes to one element update the other. You can link a spreadsheet to a part or assembly and drive the parameters from cells in the spreadsheet.

Can be defined to relate dimensions to functional requirements. For example, the cross-sectional area of a part can be defined with specific proportions and able to withstand a certain load (Area = Load/Material_Strength*Factor_of_Safety).

parametric dimension

Dimensions are constraints that control sketch size. The sketch geometry resizes when you change the dimension value. Together, geometric constraints and dimensional constraints control the size and shape of sketches used to create features. See also geometric constraints and dimensional constraints.

parent and child relationships

In a hierarchical system, a parent object owns dependent child objects. Deleting a parent deletes dependent children objects. For example, deleting a plate also deletes the pattern of holes on the plate. Deleting a child has no effect on its parent object. A child object usually has a single parent object and may be a parent to other child objects. In the browser, a child object is indented under its parent.

part

A group of faces that define a closed volume.

part constraints

Geometric constraints that control the shapes and relationships among sketch curves. See also geometric constraints.

part features

Parametric building blocks that are combined to create a part. Features can be sketched, placed, and duplicated. Relationships between features are maintained by geometric and dimensional constraints.

part model

A collection of geometrically and dimensionally related features that represent a physical object. A part file contains a single part. If the part was created in another CAD system, the part is a single solid with no parametric relationships among its geometric elements. Parts may originate as .SAT files, Mechanical Desktop parts, OLE objects, design (catalog) elements, or custom parts. Custom parts can be created in part files, or in place in assembly files. Parts are combined to form assemblies. In an assembly, parts can be created in relation to the geometry and topology of parts already in place.

participant component

An assembly component that is affected by an assembly feature. Participant components are listed under the related assembly feature in the assembly browser.

parting line

A work plane or sketched line, arc, or spline that shows where to split a face or part.

parts list

In an assembly, a listing of components. Usually, a parts list is single-level and consists of an item number or other designation, part name and quantity. In some cases, parts list is a term used interchangeably with bill of material, although large organizations usually make a distinction.

path

The trajectory of a sweep feature. A path can be an open or closed loop consisting of lines, arcs, ellipses, or circles, with a specified start point.

path file

See project file.

pattern element

In an assembly pattern, symmetrically arranged components. Each element contains one or more instances (individual parts). In the assembly browser, pattern elements are listed in a numbered sequence below the assembly pattern object. Each element can be expanded to show the instances in the element.

pattern feature

Multiple instances of a placed or sketched feature arrayed in a specified pattern. Patterns are defined by type (rectangular or circular), orientation, number of features, and spacing between features.

pattern instance

In an assembly pattern, the part or parts selected as an element. The elements (one or more parts) are repeated in a symmetrical pattern, each containing the same part instances. In the assembly browser, instances are nested below each pattern element.

perpendicular constraint

A geometric constraint that causes two lines or ellipse axes to lie at right angles to one another.

persistent dimensions

The function that automatically creates and places sketch dimensions when using Dynamic Input (Heads-Up Display).

perspective camera mode

A display mode in which a model displays in three-point perspective, like the way objects in the real world are perceived by the human eye.

physical properties

Physical properties of a part, including mass, radii of gyration, volume, principal moments, products, center of gravity, and principal axes. Mass properties are calculated with respect to the sketch coordinate system and the density assigned to an object through a material property attribute.

placed component

A part or subassembly created in a separate file, then selected and placed in an assembly. One or more copies (occurrences) of the component can be placed from one part or assembly.

placed feature

A feature that consists of a defined mechanical shape that serves a known engineering function in a part or assembly. Examples are holes, chamfers, fillets, shells, face drafts, and plane cuts.

planar face

A 2-dimensional (flat) part face.

point

A persistent sketch point, work point, curve endpoint, midpoint, or center point.

pointer input

Displays the starting point for a sketch element as X and Y Cartesian coordinates in a tooltip near the cursor when Dynamic Input (Heads-Up Display) is enabled.

post-weld machining

In weldment design, a machining operation that is performed after welds were applied to the weldment. Machining operations often affect multiple assembly components. Holes and extrude cuts are typical post-weld machining features.

See also machining features and machining features group.

preparation features

Assembly features added to the Preparations folder in a weldment assembly. Preparation features contribute to an optional metal removal process often called a weld prep. They are added before welding to ensure adequate weld strength. Chamfers are typical weld preparations. See also weld prep.

preparations feature group

A folder in the weldment environment representing an optional metal removal process. This process is done before welding to ensure adequate weld strength. The removed metal is typically filled back in with the weld bead. Chamfers are typical weld preparations. See also weld prep.

presentation view

A specialized assembly view. You develop the specialized views, such as exploded views, in a presentation file and then use them to create drawing views or other presentations.

primary entity

Any line or arc with both of its endpoints in the selection set.

principal curvatures

The principal curvatures of a surface at a point are the minimum and maximum of the normal curvatures at that point. (Normal curvatures are the curvatures of curves on the surface that are lying in planes, including the tangent vector at the given point.) The principal curvatures are used to compute the Gaussian and Mean curvatures of the surface.

profile

A closed loop defined by sketched or reference geometry that represents a cross section of a feature. An open profile defined by sketched segments, arcs, or splines can define a surface shape or extend to boundaries to close a region. A profile can enclose islands.

project

A means to organize Autodesk Inventor files and maintain valid links logically between files. A project consists of a home folder, a project file that specifies the paths to the locations of the files in the project, and the local and network folders containing Autodesk Inventor files.

You can have as many projects as needed to manage your work. The project file for each project must be maintained in the project home folder.

project file

A file that specifies the locations of files in a project. A project file is a text file with an .ipj extension. We recommend that the file is maintained in the home folder for the project. Specify the paths to all files to manage the links and references between the Autodesk Inventor files in the project.

There are four types of locations that can be specified in the project file: workspace, local search paths, work group paths, and libraries.

For most projects, a single work group location is sufficient.

Library folders are for any released or read-only parts referenced by the project, but not created or edited by it. A project can reference documents from several different library folders.

Workspaces are used where the Multi-user mode is set to Semi-isolated. Creating a project file with the work group and library locations set to reference shared folders using UNC paths is recommended.

If you work on collaborative projects, the project files can specify many file locations and even reference a secondary project file.

projected geometry

Geometry (model edges, vertices, work axes, work points, or other sketch geometry) projected onto the active sketch plane as reference geometry. Can include edges of a selected assembly component that intersects the sketch plane when it was cut in an assembly cross section.

promote

In the browser hierarchy, dragging a feature or component closer to the top level.

promote surfaces

Imported surfaces are placed in the construction environment where commands correct modeling defects such as gaps between surfaces. When corrected, surfaces are promoted to the part environment where they can be edited with feature commands.

property

A characteristic of a Microsoft Windows file that can be manipulated from an application or Microsoft Windows Explorer. Properties include author or designer and creation date and can also be unique properties assigned by applications or users. Specifying properties can be useful when searching for part or assembly files.

pull direction

The direction a mold is pulled from a molded or cast part. The definition of draft for part faces includes the pull direction.

quilt

A group of connected faces that do not define a volume.

radial dimension

A dimension that controls the distance from the center point of a circle and arc to a point on its circumference. Can be stated as a numeric constant, as a variable in an equation, or in a link to a parameter file. In drawing views, can be designated as a driving dimension (resizes the model) or reference dimension (does not resize model).

radius

Distance from the center point of an arc or circle to a point on its circumference.

rail

A structural member typically of round tube stock that often is placed around platforms for safety reasons. Typical rail frames include hand rails, vertical rail posts, and one or more intermediate/horizontal safety rails.

range

A limit placed on a variable. When a range is specified, the value of the variable must lie between the minimum and maximum of the range.

rapid prototype

The automatic construction of physical objects using additive manufacturing technology. A machine reads in data from a CAD drawing and lays down and fuses successive layers of liquid, powder, or sheet material to build up the model from a series of cross sections.

reattach sketch

Moves a sketch and attaches it to a different sketch plane than the one on which it originated. Use constraints and dimensions to position it and modify sketch size as needed.

redundant constraint

Constraints placed on fixed topologies that do not remove degrees of freedom, but do not conflict with previously placed constraints. It is marked in the browser with a warning icon, but is allowed to exist because it causes no harm.

reference dimension

In drawing views, a dimension that does not control the model size and is not intended for inspection.

reference edge

A sketch curve created by projecting a model edge onto the sketch plane. The curve can be a line, arc, circle, elliptical arc, or spline.

reference geometry

Sketch geometry created by projecting the edges, vertices, or work features of another sketch onto the active sketch plane or onto the edges of a face used to define the sketch plane. Reference geometry can be used to constrain sketch geometry or included in a profile or path. Curves that represent the boundary edges of a face used to create the sketch plane cannot be deleted or trimmed, but projected curves can be deleted or trimmed.

reference part

A simplified part that cannot be edited in Autodesk Inventor. An example is a part imported from another CAD system in .sat or .stp format.

reference sketch

In an assembly, geometry that exists on one part can be projected onto the sketch plane of a new part. The resulting cross-part sketch geometry is a reference sketch. The size and position of the reference sketch is based on the parent part. A reference sketch can be used like any other sketch geometry to create a feature in the new part.

reference sketch geometry

Geometry derived from and associated with sketch geometry from other unconsumed sketches.

reference vertex

A sketch point generated by projecting a model vertex onto the sketch plane.

reference work axis

An unbounded sketch line generated by projecting a work axis onto the sketch plane.

reference work point

A sketch point generated by projecting a work point onto the sketch plane.

refresh

A file menu command. Replaces any file in session that has outdated edits compared to the saved version of the file on disk. All components and their dependents are reloaded without closing the current assembly. If any pending edits exist for a file needing refresh, a prompt appears enabling Save Copy As to preserve the changes before conducting the refresh operation.

region

A closed profile on a face or plane defined by sketched curves, edges, or planes. An extruded region results in a solid body.

restored pattern element

A new pattern element associated with a previously created pattern. A restored pattern element replaces a pattern element whose link to the pattern was severed (made independent). In the browser, the restored element pattern is listed below other pattern elements and has a numeric reference to the previously severed element.

restructure assembly hierarchy

In the assembly browser, components promoted or demoted in the hierarchy or moved from one assembly to another assembly. Components do not move relative to other components in the graphics window when position is changed in the assembly browser. Constraints between components are retained.

revolved feature

A solid feature created by revolving a profile around an axis. Unless the revolved feature is the first feature, its relationship to an existing feature or body is defined by selecting a Boolean operation (join, cut, or intersect), and the participating bodies for the operation if multiple bodies exist. Optionally, can create a new body.

rigid body

A component that cannot change shape or size and acts as a single unit when moved, constrained, or rotated. For example, a subassembly is a rigid body because its shape and size cannot change. Although it contains multiple parts, a subassembly behaves as a single component when placed in an assembly.

rotation constraint

A motion constraint that specifies rotation of one part relative to another part using a specified ratio. Used to specify motion of gears and pulleys, for example.

rotation-translation constraint

A motion constraint that specifies rotation of one part relative to translation of a second part. Used to specify motion of a rack and pinion, for example.

rounds

Fillets created on convex edges.

same X coordinate

A Mechanical Desktop constraint replaced by the vertical constraint in Autodesk Inventor.

same Y coordinate

A Mechanical Desktop constraint replaced by the horizontal constraint in Autodesk Inventor.

SAT file

The native file format (ASCII) produced by the ACIS geometry-modeling kernel.

scene

A complete definition of components, such as geometry, materials for that geometry, lighting, a studio, and a viewpoint to create a realistic image.

search paths

The file locations specified in a project. When you open a file, Autodesk Inventor searches all the paths specified in the active project to find any referenced files.

secondary entity

Any line or arc that has only one of its endpoints in the selection set.

section

In an assembly, a view of the model defined by temporarily hiding portions of components or features on one side of a specified cutting plane.

section plane

In a base body, a cross section that defines the plane about which the body is extended or contracted.

selection tags

Labels that appear in the in-canvas display and prompt for the selection of profiles, faces, and axes for feature creation and editing.

selection window

Using the Select command to size a window, selects geometry partially or enclosed in the window.

sequence

A group of tweaks in the animation of an exploded assembly. Each task in an animation consists of one or more sequences, which in turn consists of one or more tweaks.

shape point

An internal point that defines the shape of a spline. Shape points can move when a spline is solved. By constraining a shape point to other geometry, it changes to a fit point.

shared sketch

A sketch used by more than one feature; for example, a sketch containing hole centers for different hole features.

sheet metal bend plate

In sheet metal design, that portion of the (flat or folded) model that contains the material that deforms during a bend or fold.

sheet metal design environment

Entered by converting a part to sheet metal or specifying sheet metal material for a new part. Modeling and flat pattern operations are specific to sheet metal design.

sheet metal edge

In sheet metal design, the thin face (material thickness) at the edge of a part. In part modeling, edges are the boundary between faces on the solid model.See also: sheet metal plate detail face.

sheet metal face

In sheet metal design, faces (the sketched outline of material added to a part) are planar and have a uniform thickness defined by the reserved sheet metal parameter *Thickness*. In part modeling, faces have no thickness and are bounded by edges.

sheet metal plate

In sheet metal design, plate can be used synonymously with face.

sheet metal plate detail face

In sheet metal design, the plate detail face is defined as the face of the edge of the sheet. For rectangular detail faces, one of the side lengths will equal the reserved sheet metal parameter *Thickness*. The plate detail face is referred to during the definition of unfold equations.

sheet sketch

A sheet sketch is associated to the underlying sheet and is created if no drawing view is selected when entering the Sketch environment.

Note: If a drawing view is selected when entering Sketch environment, the sketch and sketch geometry are associated to the view. It is referred to as a view sketch.

sheets

Corresponds to a page that contains drawing views. Each sheet can contain a sketch overlay that contains customized sketches that correspond to the drawing views.

shell

A parametric feature used most frequently for cast or molded parts. From a specified face, material is removed from the part, leaving a cavity with walls of a specified thickness. Shells usually have walls of uniform thickness, but individual faces can be selected and their thickness is specified. Shell walls can be offset to the inside, outside, or both sides of the part, relative to the original part surfaces.

single plane projection

A view angle where only one of the three fundamental model planes (X, Y, or Z) is visible and parallel to the view plane. Only model surfaces that are exposed in this view state are visible.

skeleton

In Autodesk Inventor, one or more parts, contained in an *.ipt or *.iam file, that are based on solids, 2D sketches, 3D sketches, or surfaces, and represent the structure of a design. Edits to a skeleton produce design changes on all associated geometry and designs.

skeleton part

In Frame Generator, a model which collects references in an *.ipt or *.iam file that are used to build a structural frame. References can include solids, 2D or 3D sketches, and work geometry. Edits to a skeleton part produce design changes on all associated geometry and designs.

sketch

A sketch consists of the sketch plane, a coordinate system, 2D curves, and the dimensions and constraints applied to the curves. A sketch can also incorporate construction geometry or reference geometry. Sketches are used to define feature profiles and paths.

sketch constraints

See constraints.

sketch geometry

Curves that occur in sketches. Can include lines, points, rectangles, splines, fillets, arcs, circles, and ellipses.

sketch geometry in drawings

A drawing sketch is an overlay that can contain text and 2D geometry, such as lines and arcs.

sketch plane

A planar face or work plane on which the current sketch is created.

sketched feature

A feature that originates from a 2D sketch. Extruding, sweeping, revolving, or lofting a sketched profile creates feature volume. The volume created by the sketched feature can be joined with, cut from, or defined where it intersects with the volume of an existing feature.

sketched part

Originates from a custom sketch rather than from an imported solid or predefined shape.

sketched symbol

A custom symbol you create and save as a Drawing Resource in a drawing or drawing template. A sketched symbol can include sketched geometry, linked or embedded bitmaps, and text. It can also contain data translated from DWG files.

You can add a sketched symbol to a drawing sheet as an annotation.

slice graphics

Reveals hidden plane or face by temporarily slicing away a specified portion of a model. The revealed plane can be used as a sketch plane.

solid body

An enclosed 3D body that has volume.

solid model

A model in SAT or STEP file format imported in Autodesk Inventor as a base feature (the first feature in a model). A base solid is fully dimensioned. Its size can be changed only by commands in the Solids environment, but the solid can be modified by adding features.

solid weld bead

In a weldment model, an assembly feature that creates the actual weld bead geometry. A solid weld bead adds mass to the assembly and can be used in interference checking. Solid weld beads reside only in the Welds feature group. Solid welds are not shown as participant components in assembly feature participation lists.

spline

A smooth curve fitted to a sequence of points within a specified tolerance.

split

Split Face divides one or more faces at a specified parting line. Resulting faces can have individual face draft applied. Trim Solid removes a section of the part. Split Solid creates a multi-body part by dividing a solid object into two separate bodies. Resulting bodies can have unique features that are not shared with other bodies.

stairway

Platform frames often exist on multiple levels surrounding large equipment, these levels often require stairway access. Stairways typically consist of C channel structural members in parallel on an incline with one or more stair treads depending on the rise required.

standard iParts

An iPart factory generates iParts, each of which has a unique configuration whose parameters, properties, and other values are maintained in an embedded spreadsheet. Each row in the spreadsheet represents an individual iPart with a unique configuration. When a standard iPart is used in an assembly, its values cannot be modified.

standard spline fit method

Generates a curve with smoother continuity than the spline-fitting method in AutoCAD.

static drawing

A drawing file on which updates are deferred. A static drawing does not update when the model changes and you cannot place views or make other changes to the drawing data. The defer updates setting is on Drawing tab of the Document Settings dialog box.

STEP file

An international format developed to overcome some of the limitations of current data conversion standards. Files created in other CAD systems can be converted to STEP format and imported in Autodesk Inventor.

stretchy feature

See adaptive feature.

structural member

A single piece of material used in the construction of a frame, typically purchased in stock lengths and sizes. Common structural members include steel angle iron, round/square/rectangular steel tubing, c-channel, i-beams, and so on.

style

In assemblies, a style is the set of information that describes component appearance. A style includes descriptors such as color diffusion, specular color, ambient color, emissive color, opacity, shine, and material.

style library

Defined styles reside in the style library and are referenced in documents. Styles in documents (local styles) can be modified as needed, but the original style definition in the library is not affected.

subassembly

An assembly file used in another assembly. The subassembly behaves as a single unit, such as a motor with a gear reducer. Parts can originate in part files, as OLE objects, or as iFeatures, or can be imported from the Mechanical Desktop or other CAD system.

suppressed assembly pattern element

A member of an assembly pattern whose geometry no longer appears in the graphics window. You can suppress an assembly pattern element if it would interfere with other components such as a rod, notch, fastener, or other geometry that would interrupt the pattern. In the browser, a suppressed element is indicated by a shaded symbol and strike-through text. A suppressed assembly pattern element can be unsuppressed.

suppressed feature

A feature whose geometry no longer appears in the graphics window. Any features dependent on a suppressed feature are also suppressed. Suppressed features are indicated in the browser by a shaded icon. A feature can be unsuppressed. It and its dependent features are visible in the graphics window and can be selected for editing.

suppressed sketch pattern element

A member of a sketch pattern whose geometry is no longer available in a profile and does not appear in drawing sketches. In the graphics window, a suppressed pattern element is shown in a dashed line. You can suppress a sketch pattern element if the pattern would interfere with other model geometry that would interrupt the pattern.

surface

A 2-dimensional definition of forms and shapes or an object in 3-dimensional space that defines area without volume.

surface normal

Solid bodies are bounded by surfaces. The surface normal is the outward-pointing vector perpendicular to a particular point on a surface. When assembly constraints are being placed, a surface normal vector appears. In most cases, it can be flipped in the opposite direction.

sweep feature

A feature created by moving a profile along a path. A sweep feature usually requires two sketches, a profile, and a path on intersecting planes. An optional guide rail for scaling can be included in a third sketch. Unless the sweep feature is the first feature, its relationship to an existing feature or body is defined by selecting a Boolean operation (join, cut, or intersect), and the participating bodies for the operation if multiple bodies exist. Optionally, can create a new solid body.

tangent constraint

A geometric constraint that causes two curves to have the same slope at the point where they intersect. For example, a line can be tangent to an arc, circle, or ellipse, but two lines cannot be tangent to one another.

In assemblies, a tangent constraint can be applied between cylindrical, conical, and toroidal faces or circular arc edges. On selected components, one component moves toward another, and contacts at the point of tangency. In assemblies, tangency can be inside or outside a curve, depending on the direction of the selected surface normal.

taper

Sets positive or negative taper angle for sweeps, extrusions, and coils normal to the sketch plane.

target point

A point in space that your line of sight passes through. In orthographic camera mode, the point can be located on the X and Y axes. In perspective camera mode, the point can be located on the X, Y, and Z axes. The target point also serves as a center point for the Rotate command.

task

An assembly or disassembly action in the animation of an assembly explosion. Each task in an animation consists of one or more sequences.

template

An assembly, part, or drawing file that contains predefined file properties. To create a file based on a template, you open a template file, create the content, and then save it with a unique file name. Pre-defined properties can include visible default reference planes, customized grid settings, color scheme, drafting standards, and so on.

termination

The method selected to end an extruded, swept, revolved, or lofted feature. Examples are fixed length or angle, extension through all faces or to the next face, or equal extension from a selected plane.

title block

The area on a drawing sheet that identifies the owner, includes a description of the drawing, and provides other relevant information.

toolbody

An intermediate volume produced when a swept, extruded, or revolved feature is created.

top-down design

An assembly technique in which parts are designed in place within an incomplete assembly, taking advantage of existing part placement and using part features and positional information as the basis for the design or modification of new parts. Parts designed in the assembly context are designed relative to features on parts already in the assembly.

top-level assembly

The root of an assembly, under which all components are arranged in a hierarchical structure, is automatically created when you create an assembly file. In the browser, the top level of the assembly is represented by an icon and by default, the file name. In an assembly file, double-click the top level of the assembly to switch from part creation or editing (the part environment) to assembly tasks (the assembly environment).

trails

Lines in an exploded view that show the relationship of a component to the assembly. Trails indicate the direction and distance that a component was moved to create the view.

transform

Change in position or direction or orientation, such as move or rotate.

triad

A 3D coordinate display where the X axis is red, the Y axis is green, and the Z axis is blue. The triad is used to:

1) locate line segments in a 3D sketch.

2) move and rotate grounded work points and faces.

3) tweak components in a presentation file.

4) create a UCS (User Coordinate System).

5) display the coordinate system in both 2D and 3D sketches.

tweaking

The process of moving a component in an assembly presentation to create an exploded view.

tweaks

Adjustments made to assembly components in an exploded view. You specify the distance and direction of movement to provide better visibility of the components and their relationships.

type of draft

Face draft may be shadow and edge draft. Shadow draft selects continuous tangent faces and drafts all edges in one operation. Edge draft is draft applied to a single face that is not tangent to another face. Face draft is shown in the browser as TaperShadow or TaperEdge.

unavailable geometry

See missing geometry.

unconsumed sketch

A sketch in a part or assembly model that was not used in a feature. An unconsumed sketch can be used to show assembly layout and develop design concepts. You can display unconsumed sketches in drawing views.

undefined geometry

See missing geometry.

underconstrained geometry

Features or parts with unsolved degrees of freedom are underconstrained. Features and parts designated as adaptive have underconstrained geometry that resizes when it is constrained to fixed geometry (all degrees of freedom removed).

undisplayed part

A part or subassembly whose visibility was turned off in the assembly context. Such parts are not opened or loaded into memory. Parts or subassemblies are turned off when they are not needed for the current modeling tasks.

update

A part or assembly is recalculated after significant changes. Update can be triggered automatically or manually, allowing work to continue before a part or assembly is updated. An update incorporates changes held in memory from the current editing session and updates the display in the graphics window and browser. Unlike Refresh, Update incorporates only edits made to the active component, but does not retrieve the saved version from disk.

update all

Recalculates all components to incorporate changes held in memory, including the top-level assembly. Update All can be triggered automatically or manually, allowing work to continue before a part or assembly is updated. Unlike Refresh, Update All incorporates only edits made locally, but does not retrieve the saved version from disk.

upstream process

An operation that precedes the selected operation in an order-dependent process. For example, weldment preparations are upstream from the weld bead and post-weld machining operations.

value input box

The value input box is used to enter numeric values for Direct Manipulation modeling and editing operations. It is located just above the mini-toolbar in the in-canvas display.

variable-radius fillet

A fillet that has a radius that varies along its length. You set a different radius for the start point and endpoint. The transition type determines the shape of the fillet. Use the Variable tab on the Fillet feature dialog box to select the edges and specify the radii for a variable-radius fillet feature.

vertex

The bounding point of two or more curves or model edges.

vertical constraint

A geometric constraint that positions selected lines, ellipse axes, or pairs of points parallel to the Y-axis of the sketch coordinate system (same X coordinate).

vertical dimension

A dimension that controls the size of curves that are parallel to the Y axis in sketches and drawing views. Can be stated as a numeric constant, as a variable in an equation, or in a link to a parameter file. In drawing views, can be designated as a driving dimension (resizes the model) or reference dimension (does not resize the model).

view sketch

If a drawing view is selected when entering Sketch environment, the sketch and sketch geometry are associated to the view. It is referred to as a view sketch.

Note: A sheet sketch is associated to the underlying sheet and is created when no drawing view is selected when entering the Sketch environment.

visibility

A characteristic of an assembly component that determines whether it appears in the graphics window. In large assemblies, it is useful to turn off the visibility of components not needed in the current design.

void

A group of faces that define an internal hollow space. For example, when a cube is shelled without removing a face, the result is a void inside the cube.

weld bead feature

A weldment assembly feature type used to create complete weld specifications in the model. Weld bead features, whether cosmetic or solid, reside only in the Welds feature group folder. Weld bead features are not shown as participants in assembly feature participation lists. See also cosmetic welds and solid welds.

weld feature groups

Folders in the weldment browser used to organize assembly features that combine to create a weldment. Three weld feature groups exist for weldments: Preparations, Welds, and Machining. Each group represents a particular task in the manufacturing process and is represented in the browser with a different icon.

weld prep

An optional metal removal process done before welding to ensure adequate weld strength. The metal removed is typically filled back in with the weld bead. Chamfers are typical weld preps. See also preparations feature group and preparation features.

weldment design environment

A specialized assembly environment that is accessed by converting an assembly to a weldment or by selecting a weldment template for a new assembly. The weldment environment provides all the design commands used in the assembly environment along with unique commands for defining weldments.

welds feature group

A group in the weldment browser that represents the process of adding weld bead features to a design. Welds are applied after optional weld preparations and before post-weld machining operations. Weld features include cosmetic or solid fillet weld beads.

work axis

A construction feature that defines an axis in 3D space. A work axis is useful when no geometry exists to use as an axis. Work axes can be projected onto a sketch and incorporated into dimension and constraint schemes.

work feature

Plane, axis, or point that can be projected onto a sketch as a reference feature and used to construct new features. A work feature can be incorporated into dimension and constraint schemes, but is not model geometry.

work plane

A construction feature that defines the parametric location of a sketch plane in 3D space. A work plane is useful when no planar face exists to use as a sketch plane, for example, when sketching on curved or toroidal faces. A work plane can be incorporated into dimension and constraint schemes.

work point

A construction feature that defines a point in 3D space. A work point can be projected onto a sketch and incorporated into dimension and constraint schemes.

workgroup search paths

The shared network locations that contain the Autodesk Inventor files for your design team. Each project specifies the locations of workgroup search paths.

workspace

The path to the work area on your local machine. Your workspace is specified in the active project.

x-ray ground shadow

Shadow cast below a visible model is updated dynamically as the model rotates or moves. Details of internal geometry are distinguishable. Effect is like a shadow cast on the ground beneath a model.