Glossary

A

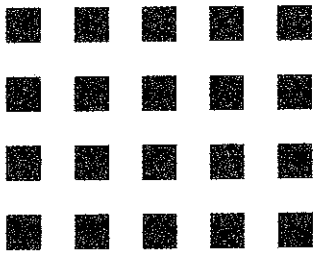
active constraints Constraints that have been applied to geometry. (Ch. 5)

adaptive assemblies Assemblies created using the adaptive design approach. (Ch. 12)

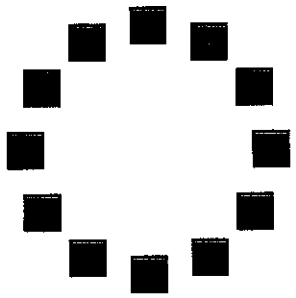
adaptive design A design approach in which features and/or parts are intentionally underconstrained when they are created. Assembly constraints are later added to finish defining and constraining them. (Ch. 12)

American National Standards Institute (ANSI) The foremost producer of standards in the United States. ANSI standards are the most commonly used in design and drafting in the United States. (Ch. 8)

arrays Patterns of duplicated features. Inventor allows you to create both rectangular and polar (circular) arrays. (Ch. 10)



Rectangular Array

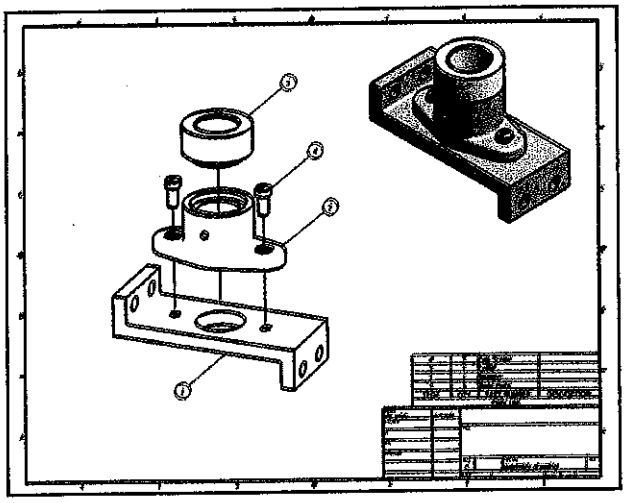


Polar (Circular) Array

assembly-based fit function Inventor's ability to fit parts based on assembly or design intent, which allows users to concentrate on the design without depending on the associated parameters or constraints. (Ch. 1)

assembly constraints Constraints that control the relationships between parts in an assembly drawing. (Ch. 12)

assembly drawings 2D drawings that provide information about part identification within an assembly. (Ch. 14)



Assembly Drawing

assembly modeler The mode in Inventor in which assembly models are created. (Ch. 12)

assembly working drawing An assembly drawing that contains complete manufacturing information and can be used as a working drawing. (Ch. 14)

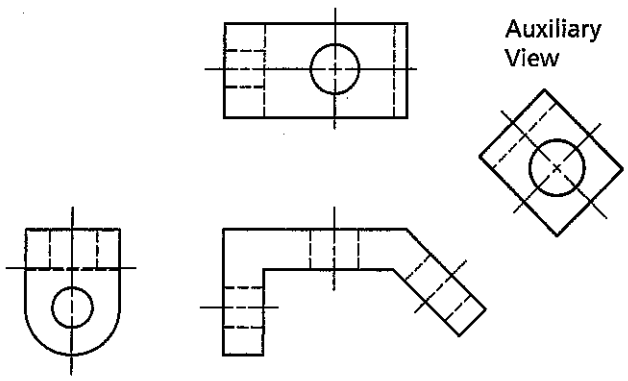
associative functionality The ability to change a design at any level, in any mode (drawing, part modeling, assembly, etc.). Any change made in any mode is automatically carried through to all modes. For example, a change made in part modeling mode is automatically reflected in any 2D drawings based on the model. (Ch. 8)

auto dimension A feature that analyzes the current sketch to see if it is fully constrained. If it is not fully constrained, this feature can add the necessary constraints automatically. (Ch. 5)

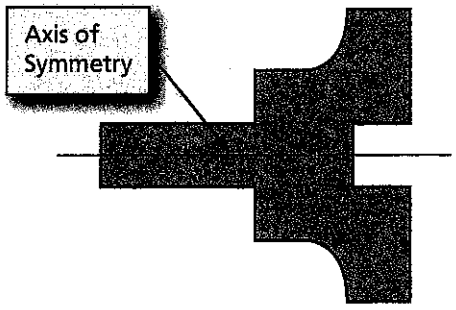
automatic exploded view An exploded view generated automatically by Inventor with little input from the user. (Ch. 13)

auxiliary plane An imaginary plane adjacent to a surface that is inclined in the regular orthographic views. (Ch. 9)

auxiliary view A 2D drawing view that has a line of sight perpendicular to a surface of an object that is inclined in all of the regular views. The purpose of the auxiliary view is to show the inclined surface at its true size and shape. (Ch. 9)

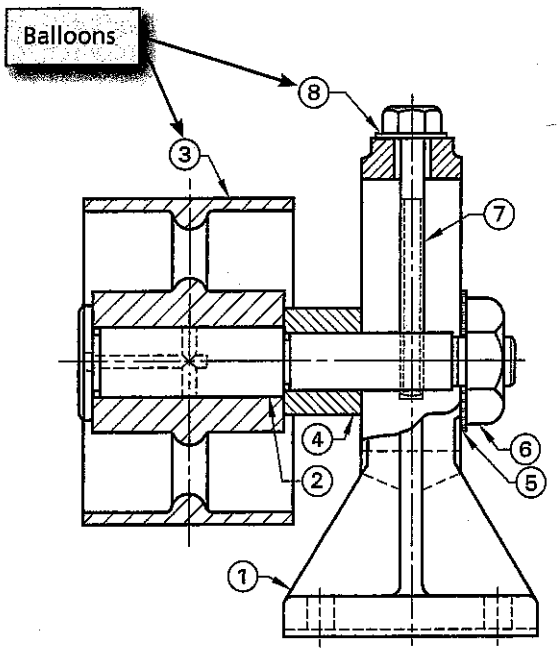


axis of symmetry A point or line that is perpendicular to the symmetrical items or to two symmetrical sides of an object. (Ch. 10)



B

balloon A small circle on an assembly drawing that contains a number that corresponds to a part number given in the parts list. The balloon is connected to the corresponding part in the assembly by a leader. (Ch. 14)



base component The first component in an assembly, on which all other components are based. (Ch. 12)

base feature The first solid feature that is built for a model. The base feature is usually the primary shape in the model. (Ch. 3)

base orphan reference node (BORN) technique A method in which a Cartesian coordinate system is the first feature. This allows the first node in the history tree to be an orphan, allowing reference geometry and the Cartesian coordinate system to be used as sketch planes. (Ch. 7)

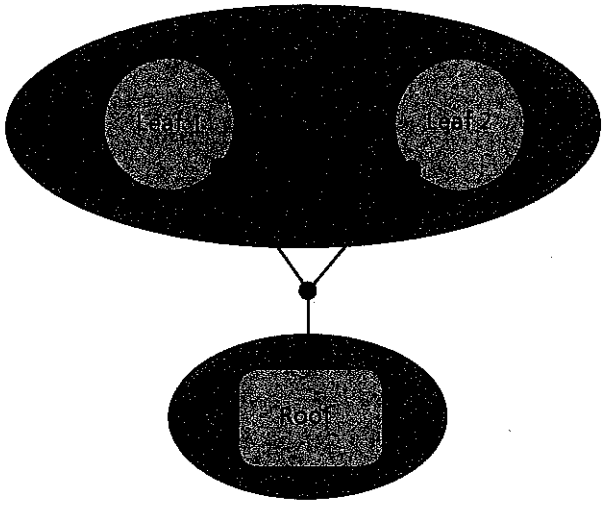
base view The first drawing view created in drawing mode, from which other views can be projected. (Ch. 8)

bidirectional associative functionality Associativity that occurs in both directions. If a part is changed in an assembly model, the corresponding part file is changed automatically to match it; if the part file itself is changed, the assembly automatically updates to match the change in the part. This bidirectional functionality applies to all modes within Inventor. (Ch. 12)

bill of materials (BOM) A table that contains information about the parts within an assembly, including information such as part names, quantities, costs, vendors, and all other information related to building the part. (Ch. 14)

BILL OF MATERIALS			
NAME	REQD	MATL	NOTES
IDLER PULLEY	1	C1	
IDLER PULLEY FRAME	1	C1	
IDLER PULLEY BUSHING	1	BRO	
IDLER PULLEY SHAFT	1	CRS	
Ø.62 HEX NUT	1		PURCHASED
WOODRUFF KEY #405	1		PURCHASED
.12 OILER	1		PURCHASED

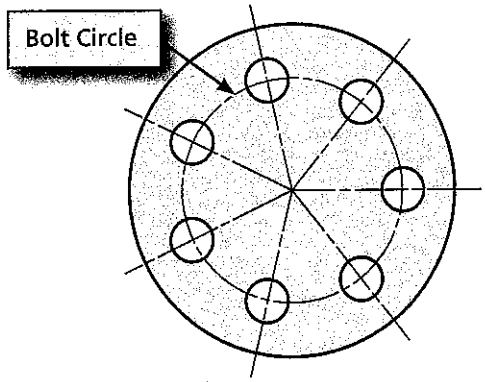
binary tree A graphic display of the history of a model, including all of the operations used to create it. (Ch. 3)



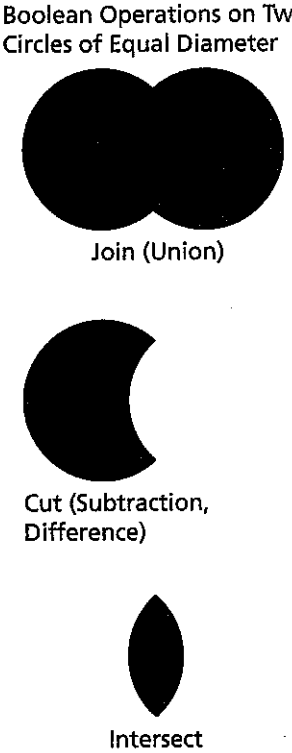
Binary Tree

bisect Divide into two equal parts.

bolt circle A circular hole pattern in which the center of each hole is at an equal distance from the center of a bolt or other circular part. The bolt circle passes through the center of each hole in the part. (Ch. 10)



Boolean operations Solid modeling operations based on Boolean mathematics. In Inventor, the three Boolean operations are Join, Cut, and Intersect. (Ch. 3)

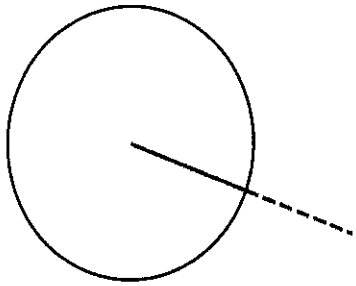


Boolean relations Features created using Boolean operations and relationships. (Ch. 4)

BORN technique *See* base orphan reference node (BORN) technique. (Ch. 7)

bottom-up approach An approach to creating an assembly drawing in which the individual parts are created first. They are then pulled together into an assembly file. (Ch. 12)

boundary When the Trim or Extend command is active, the edge that defines a boundary to which Inventor trims or extends geometry. (Ch. 6)

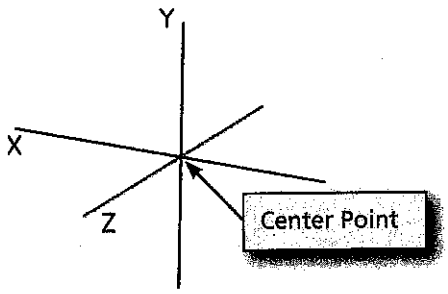


In this Trim operation, the circle acts as the boundary. Notice that Inventor dashes the part of the line to be trimmed.

boundary representation (B-rep) A method of representing objects in terms of their spatial boundaries. (Ch. 1)

C

center point The point at which the three work axes meet. This point is available by default in Inventor when the BORN technique is used. (Ch. 7)



child A feature that is created based on existing (parent) features. (Ch. 7)

composite solids Solids that have been combined using Boolean operations. (Ch. 3)

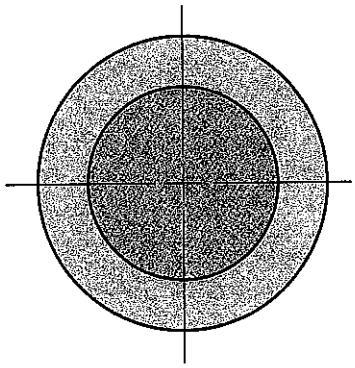
computer-aided drafting (CAD) Drafting processes in which the computer is used to create drawings that were formerly created manually, replacing instruments such as the triangles and T-squares. First-generation CAD systems were simply 2D drafting programs. Today's sophisticated CAD systems can handle 3D modeling and analysis. (Ch. 1)

computer-aided engineering (CAE) A field in which engineering design centers around one or more 3D computer models. (Ch. 1)

computer-based modeling The process of creating geometrically accurate 3D solid models on a computer. (Ch. 1)

computer numerical control (CNC) A system in which manufacturing machines and tools are controlled directly by computer programs. (Ch. 8)

concentric Sharing a common center point. (Ch. 4)



Two Concentric Circles

concurrent engineering A method of design and product development in which designers, design engineers, analysts, manufacturing engineers, and management engineers work together closely during the entire design, development, and manufacturing process. Marketing and purchasing personnel are often involved also. (Ch. 1)

constrained move The process of picking and dragging a component of an assembly model to check for constraints. The component honors assembly constraints, so it will only move in unrestricted degrees of freedom. (Ch. 12)

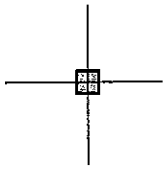
constraint network A system of appropriate geometric and dimensional constraints on 2D sketches that ensures that the geometric shape behaves predictably as changes are made in the future. (Ch. 6)

constraints Limits or restrictions that are placed on geometry to define the exact shape and size of a solid model. (Ch. 2)

construction geometry Geometry that is used to line up or define other geometry, but is not itself used as the shape geometry for the model. (Ch. 10)

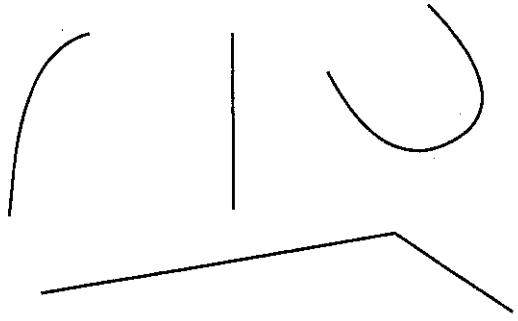
constructive solid geometry (CSG) The combination of 3D solid primitives to create an accurate 3D model. (Ch. 1)

crosshairs A precise cursor that allows you to pick a point exactly at the intersection of the horizontal and vertical lines of the cursor. (Ch. 1)



Crosshairs

curve In Inventor profiles, a basic geometric entity such as a line or arc. (Ch. 6)



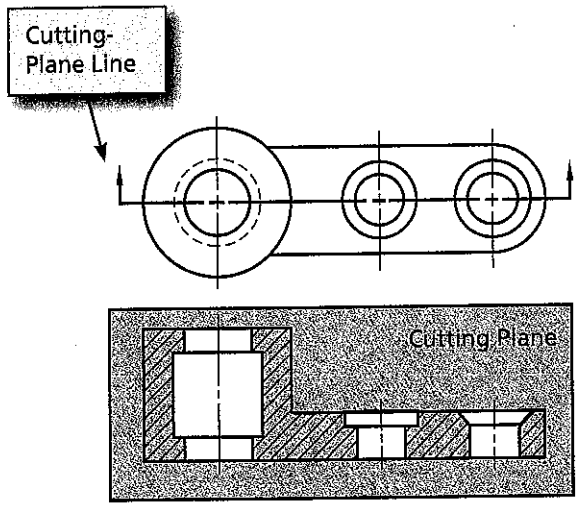
Curves

cut A Boolean operation in which the volume of one solid model is subtracted from the volume of another solid. This operation is also called a *difference* or *subtraction* operation. *See also* Boolean operations. (Ch. 3)

Cut Operation

cut feature A feature that is created expressly to be cut or subtracted from a base model. (Ch. 2)

cutting plane The path along which the section is taken for a section view. (Ch. 10)



Resulting Section View

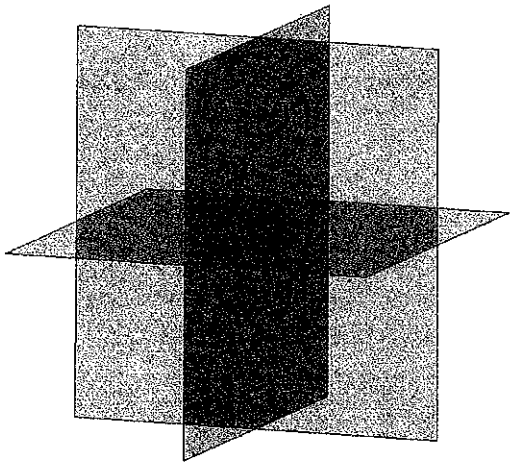
cutting-plane line The line shown in a 2D view perpendicular to the cutting plane for a section view, showing how the view transects the object being sectioned. (Ch. 10)

D

datum A point, line, or surface that is assumed to be exact. (Ch. 9)

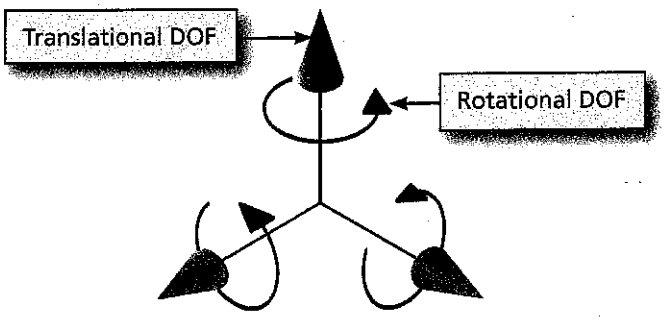
datum features User-definable work features that are updated with the part geometry. They are used to create and align features and to orient parts in an assembly. (Ch. 9)

datum planes The three mutually perpendicular planes that make up the Cartesian coordinate systems. The datum planes, or work planes, become available as sketch planes when the BORN technique is used. (Ch. 7)



The Three Datum Planes

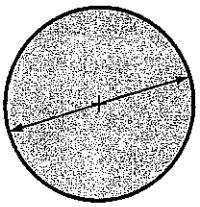
degrees of freedom The six ways in which rigid 3D bodies can move in 3D space. They include three translational degrees of freedom and three rotational degrees of freedom, based on the 3D Cartesian coordinate system. (Ch. 12)



derived dimensions Dimensions whose values are derived from and are dependent on other dimensional values. (Ch. 5)

detail drawing A drawing for a single part that includes all the dimensions, notes, and information needed to make that part. (Ch. 8)

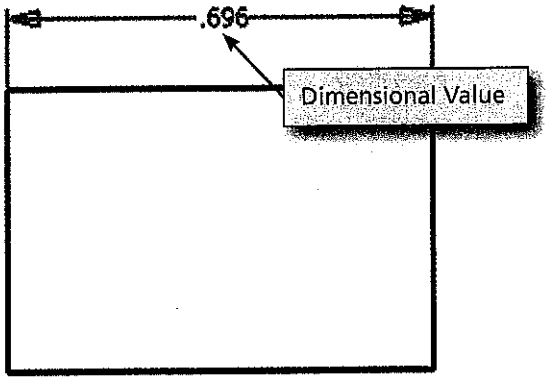
diameter The distance across a circle through its center point. (Ch. 8)



Diameter

dimensional constraints Constraints that are used to describe the size and location of individual geometric shapes. (Ch. 5)

dimensional value The actual location and size of the entity being dimensioned. (Ch. 5)



dimensional variable A dimension that is used as a control variable in a mathematical equation. (Ch. 5)

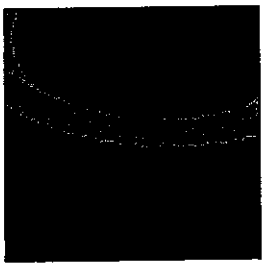
dimension format Characteristics of a dimension, including the type of units and the precision that are used. The dimension settings are stored individually with each electronic file. (Ch. 3)

dimensioning A system of describing the size and location of features on a 2D sketch or drawing. (Ch. 8)

dimension style A group of style settings that Inventor uses to display dimensions. Inventor's default dimension style cannot be changed, but the user can create and modify new dimension styles. (Ch. 8)

direct adaptive assembly An approach to building assembly models in which part relationships are defined directly and have no order dependency. (Ch. 12)

DOF symbol Degrees of freedom symbol; when turned on, the symbol that appears near each part in an assembly to indicate how many degrees of freedom are open for that part and which ones they are. (Ch. 12)



DOF Symbol

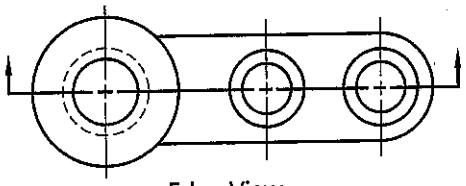
drawing template A drawing file that has been set up with specific settings and options and saved in template format so that it can be used to create new drawings. (Ch. 10)

driven dimensions Dimensions that can be added to a sketch for reference only. If added as normal (parametric) dimensions, they would over-constrain the sketch. Instead, Inventor allows you to insert them as reference dimensions whose values are driven by normal dimensions. You cannot change the parametric size or shape of a part by changing driven dimensions. (Ch. 5)

dynamic UCS A feature that aligns the current user coordinate system automatically with the sketch plane you choose when you begin a new sketch. (Ch. 3)

E

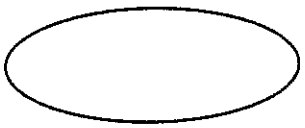
edge view A view in which a cutting plane appears as an edge, or line. (Ch. 10)



Edge View

editing-by-dragging method A method of modifying an under-constrained sketch by picking a curve or point and dragging to change the size or shape of the sketched geometry. (Ch. 6)

ellipse A regular oval that has two equal center points. (Ch. 5)



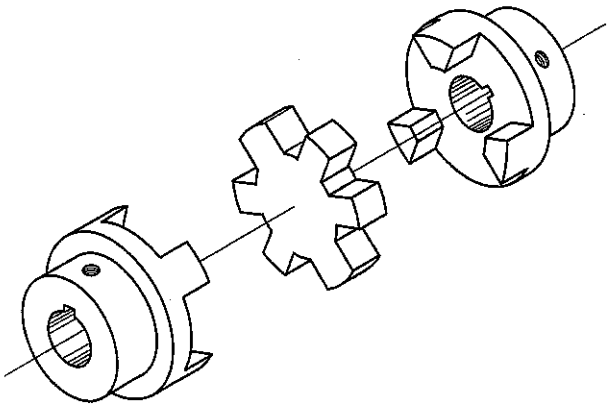
engineering drawing 2D drawing extracted from a solid model that can be printed at any exact scale. Engineering drawings are used to communicate engineering ideas and designs to manufacturing, purchasing, service, and other departments. (Ch. 8)

exit marker A horizontal arrow that displays next to the cursor during dynamic rotation; if you pick a point while the exit marker is displayed, the Rotate command ends, returning control to the active command. (Ch. 2)

Exit Marker

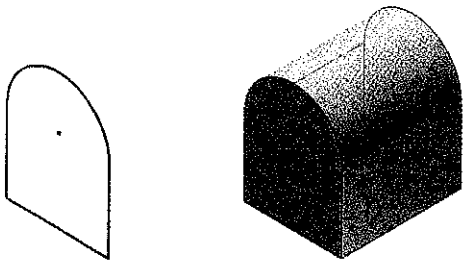
explicit relationship A parent/child relationship that is entered manually by the user. (Ch. 7)

exploded assembly An assembly model that shows all of the parts of the assembly and how they fit together. The various parts are shown in their correct alignment (as much as possible) but slightly apart so that their relationships are obvious. (Ch. 13)



Exploded Assembly Model

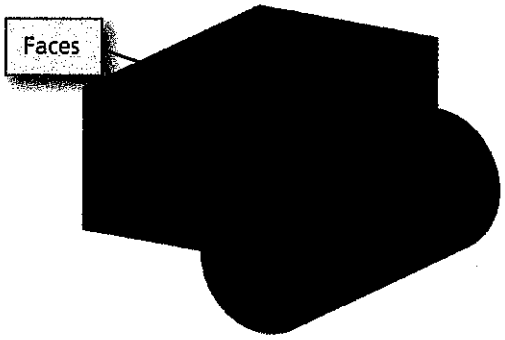
extruding Giving depth to a 2D sketch to create a 3D model. (Ch. 2)



Sketch Extrusion from Sketch

F

face A 2D surface on a surface model or solid model. (Ch. 2)



feature-based parametric modeling The process of building a solid model by adding simple features one at a time in such a way that they can be changed at any time. (Ch. 1)

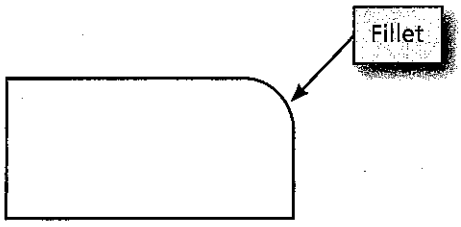
feature control variables Dimensions used as variables that control the location of related features. (Ch. 5)

feature dimensions The dimensions used to create features. (Ch. 8)

features Pre-defined parts or construction tools for which users define the key parameters. (Ch. 1)

feature suppression A method of changing a design that allows you to disable a feature while retaining all of its information in the file's database. This allows you to revert to that feature later if necessary and gives you much more flexibility to experiment, especially in the early design stages. (Ch. 7)

fillet A rounded outside corner on an object. *See also:* round. (Chs. 4, 6)

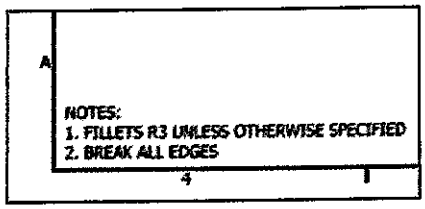


first-angle projection A method of projecting a view of a 3D object onto a 2D plane (paper) in which the object is projected from the first Cartesian quadrant. This method is commonly used in drawings based on International Standards Organization (ISO) standards. (Ch. 8)

fully constrained geometry Geometry in which constraints are used to define a model's shape, size, and location completely. (Ch. 5)

G

general drawing note A note on a drawing that applies to the entire drawing, unless otherwise specified. General drawing notes are placed by convention in the lower left area of the drawing sheet, just inside the border. (Ch. 9)



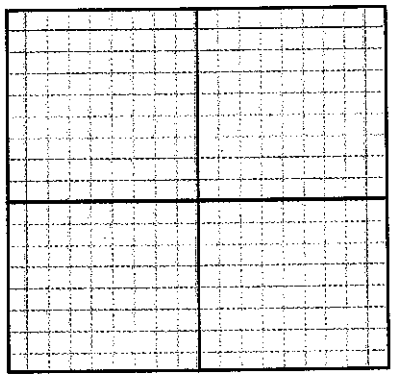
geometric constraints Restrictions that can be applied to geometric entities. Examples include horizontal, parallel, perpendicular, and tangent constraints. (Ch. 5)

geometric constraint symbol A symbol that Inventor uses to show alignments, perpendicularities, tangencies, and other relationships with existing geometry. (Ch. 2)

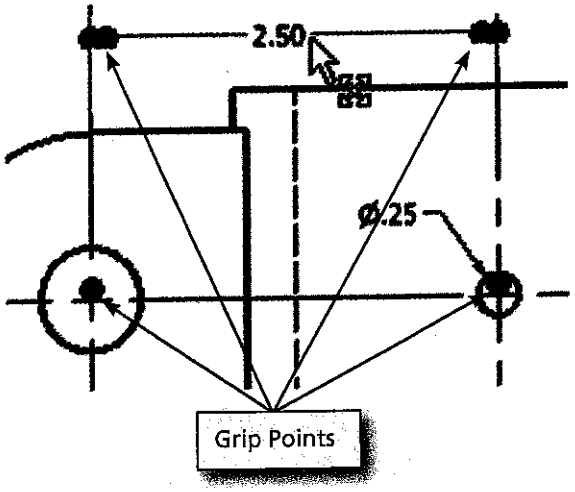
geometric construction tools Commands such as Trim and Extend that allow the user to modify 2D sketches. (Ch. 6)

geometric entities The basic geometry from which models are built, such as lines and circles. (Ch. 2)

grid A nonprinting 2D graphic that appears in sketch mode for reference. The grid is always aligned with the sketch plane. The intervals between the grid lines are customizable. (Ch. 3)



grip points The large green dots that appear at places on a dimension in a drawing file that you can click and drag to a new location. (Ch. 9)



grounded Having no degrees of freedom. The base component in an assembly is automatically grounded so that other components can be based on it. (Ch. 12)

H

half section Half of a full sectional view. (Ch. 10)

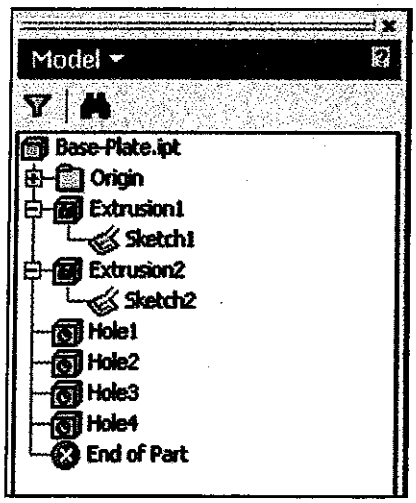
helix The curving path that a point would follow if it were to travel in an even spiral around a cylinder and parallel to the axis of that cylinder. (Ch. 12)

hidden lines Lines that would not be visible if you were looking at an object without "see-through" capability. On a 2D drawing sheet, hidden lines are shown as dashed or broken lines. (Ch. 8)

history In solid modeling, the steps and operations used to create the final model. (Ch. 3)

history-based part modifications Changes to a part that are accomplished by accessing and modifying construction steps from the model history tree in the part browser. (Ch. 4)

history tree A graphic representation of a model's history. The history tree contains a sequential record of all construction steps and their order of creation, as well as rules defining the design intent of each construction operation. (Ch. 4)



History Tree

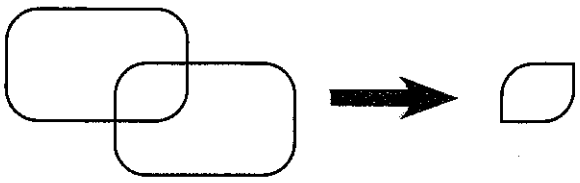
I

icons The small pictures on buttons that help identify the function of the buttons.

immediate features The direct parent/child relationships for a specific feature. (Ch. 7)

implicit relationship A parent/child relationship that is implied by feature creation. (Ch. 7)

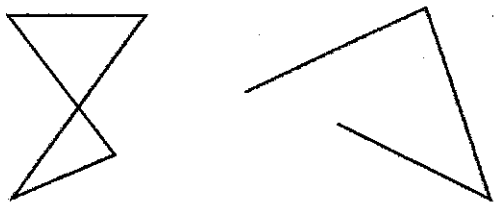
intersect A Boolean operation in which a new solid is created, consisting of only the volume that is common to all of the selected solid models. *See also* Boolean operations. (Ch. 3)



Intersect Operation

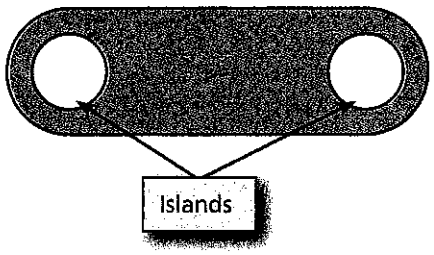
interference The overlap of two or more parts that occupy the same area in space. (Ch. 4)

invalid profile A profile that contains self-intersecting curves or open regions; such a profile is invalid because it cannot be extruded or revolved to create a 3D model. (Ch. 6)



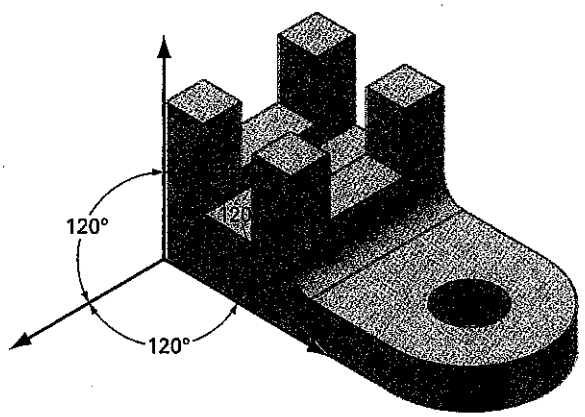
Invalid Profiles

island An area within a sketch that is not included in an extrude operation. (Ch. 2)



isometric projections Projections onto a 2D plane (paper) that are not aligned with the base view, so they can be moved freely on the drawing without affecting the position of other views. (Ch. 8)

isometric view A view that shows the model in such a way that the three coordinate axes (X, Y, and Z) all appear to be 120° apart. (Ch. 2)



Isometric View

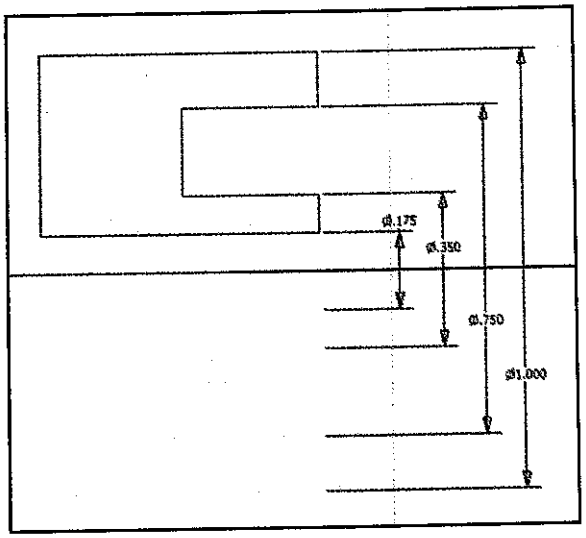
J

join A Boolean operation in Inventor that combines the entire volumes of the selected solid objects into a single solid. *See also* Boolean operations. (Ch. 3)

L

left-hand rule A rule for part rotation: If you hold the thumb, index finger, and middle finger of your left hand so that they form three right angles, the thumb points in the X direction, the index finger points in the Y direction, and the middle finger points in the Z direction. *See also:* right-hand rule. (Ch. 12)

linear diameter dimensions Dimensions that show diameter in a view that is perpendicular to the circular surface. (Ch. 10)

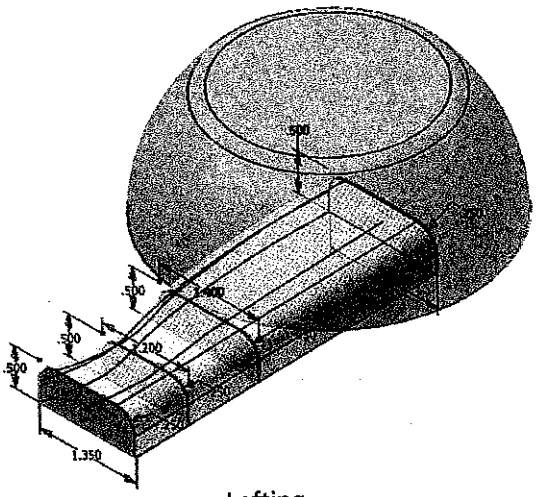


Linear Diameter Dimensions

linear sweep A specific type of sweep in which the sweep path is a straight line that is perpendicular to the profile; also called an *extrusion.* (Ch. 11)

local coordinate systems (LCS) Coordinate systems that are defined by the user and are "local"—defined only within the current electronic file. (Ch. 2)

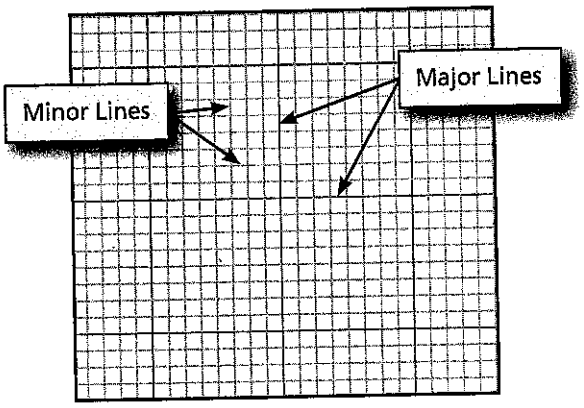
lofting A method of 3D feature creation in which multiple profiles having various shapes are blended onto separate planes to achieve a complex shape. (Ch. 11)



Lofting

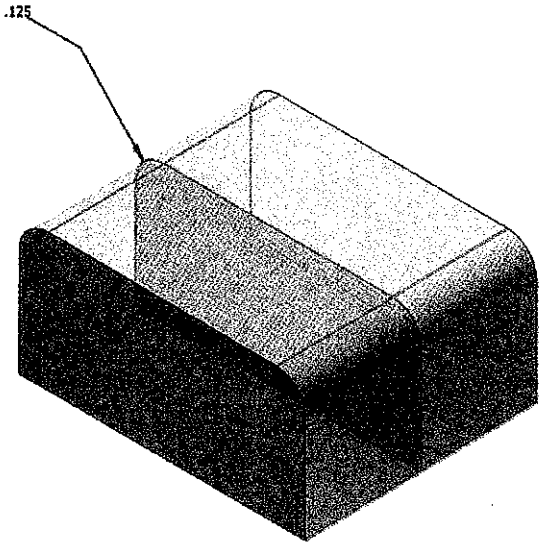
M

major lines The darker lines that appear in the grid that appears in sketch mode. (Ch. 3)



middle-out approach An approach to assembly modeling that includes characteristics of both the top-down approach and the bottom-up approach. Most of the parts are created first; then the assembly is created. Additional parts are then created as necessary using the assembly for construction information. (Ch. 12)

mid-plane extrusion An extrusion operation that extends equally in both directions from the profile. (Ch. 4)



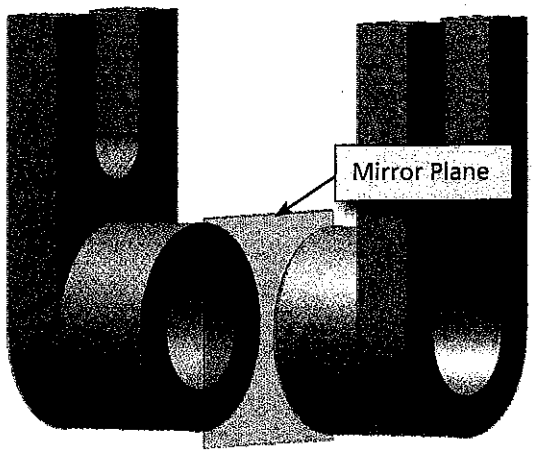
Mid-Plane Extrusion

midpoint The exact center point of an arc or line. Inventor highlights the midpoint with a green dot. (Ch. 2)

minor lines The lighter lines that appear in the grid that appears in sketch mode. There are several minor lines for every major line in the grid. (Ch. 3)

mirrored feature A feature that is created by projecting the exact mirror image of an existing feature. Mirroring a feature preserves the original parametric definitions of the feature being mirrored. (Ch. 10)

mirror plane The planar surface about which a feature is mirrored. (Ch. 10)



Mirrored Feature

model history tree Another term for the history tree. (Ch. 4)

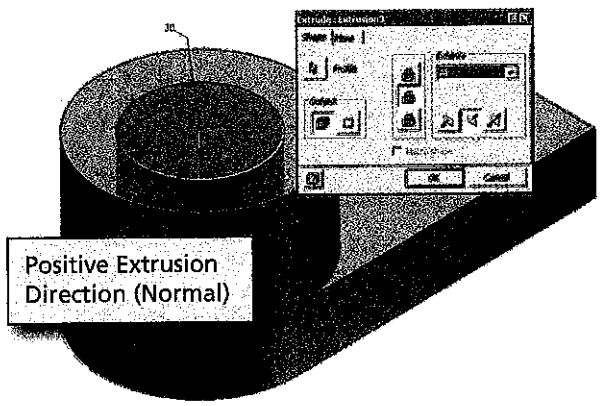
modeling strategy The number and types of features needed to construct a model and the order in which they should be created. (Ch. 3)

model parameters The parameters of a model that are created automatically when the model is created. (Ch. 5)

multiview drawings 2D drawings that contain sufficient views of a part or model to describe it completely for manufacturing purposes. (Ch. 8)

N

normal The positive side of a face, which identifies the positive extrusion direction in Inventor. (Ch. 2)



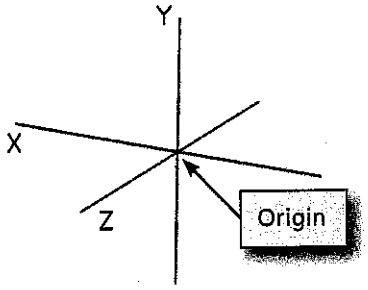
O

offset An exact replica of a profile, but at a different size. (Ch. 6)

Offset Profiles

offset work plane A work plane that is parallel to its reference plane, at a specified distance from the reference plane. (Ch. 9)

origin The point at which the axes of a 2D or 3D Cartesian coordinate system intersect. The default center point in Inventor is located at the origin. (Chs. 4, 7)

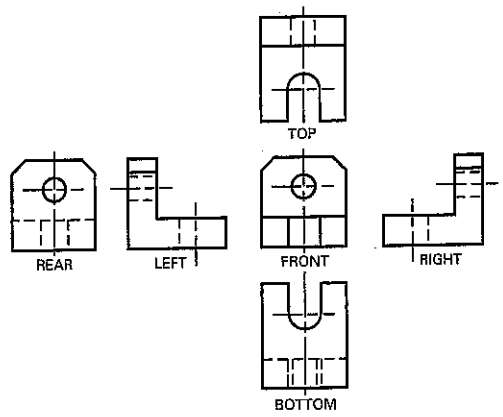


orphan The first node in a part's model history tree when the BORN technique is used. It is called an *orphan* because it is not based on any pre-existing parts, so it has no history to be replayed. (Ch. 7)

orthogonal Drawn at right angles. (Ch. 8)

orthographic projections 2D drawing views that are aligned to the base view and inherit the base view's scale and display settings. (Ch. 8)

orthographic view (1) A screen view of a model in which all of the object's edges are shown parallel to each other. *See also:* perspective view. (Ch. 2) (2) A view generated by projecting the front, top, right-side, left-side, bottom, or back of a 3D model onto a flat surface. (Ch. 9)



The Six Orthographic Views

over-constrained The condition in which a feature size or location is controlled by two separate dimensions. Inventor does not allow you to over-constrain a sketch. (Ch. 5)

P

pan Move to a different place in a graphics window at the same level of magnification. (Ch. 1)

parametric A process or design defined in such a way that it can be changed at any time in the design process by changing its geometric definitions. (Ch. 1)

parametric assembly A parametric 3D model made up of multiple part models or subassemblies. (Ch. 12)

parametric equations Equations that form the basis for parametric relations by establishing dimensions derived from other dimensional values. (Ch. 5)

parametric modeling The process of creating models using geometric definitions that can be changed at any time in the design process to change the nature of the model. (Ch. 2)

parametric relations User-defined mathematical equations composed of dimensional variables and/or design variables. (Ch. 5)

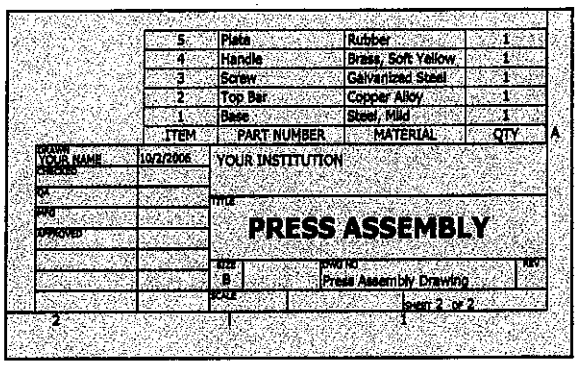
parent Any feature that is referenced during the creation of a new feature, or *child*. (Ch. 7)

parent/child relationships The relationships between new features and the features referenced during their creation. (Ch. 7)

part A sequence of engineering features that can be modified at any time. (Ch. 1)

part browser The panel on the lower left of the graphics window, which displays the model history tree. (Ch. 4)

parts list A list of parts in an assembly drawing, including details such as part numbers and quantities for each part in the assembly. A parts list is usually a subset of the bill of materials for an assembly or product. (Ch. 14)



Parts List Above Title Block

pattern leader The feature to be duplicated in a pattern or array. (Ch. 10)

perspective view A view of a model in which the model is shown with receding edges that converge toward a focal point. *See also:* orthographic view. (Ch. 2)

placed features 3D features that do not need a 2D sketch and can be created automatically. *See also:* sketched features. (Ch. 3)

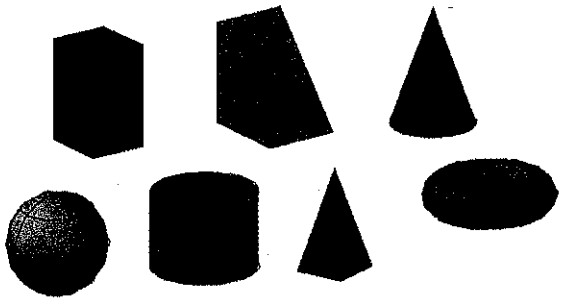
precision The number of decimal places used in a dimension. A higher number of decimal places, even if the places are occupied by zeros, denotes a higher accuracy in the finished part. (Ch. 3)

presentation background A paper-white background Inventor provides for 2D part and assembly drawings. (Ch. 13)

presentation modeler The mode in Inventor in which assembly models can be exploded and saved for presentation purposes. (Ch. 13)

primitive A basic model from which other, more complex solids can be built. Most CAD software includes primitives such as the prism, cylinder, cone, wedge, torus, and sphere. (Ch. 1)

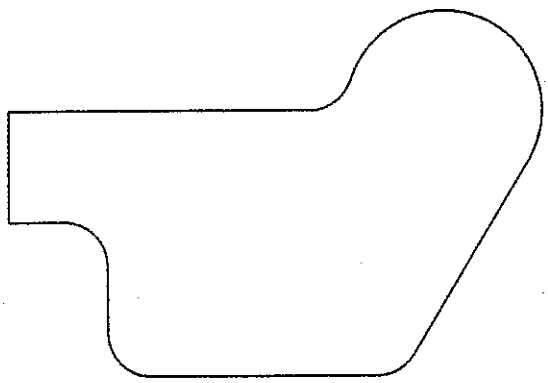
primitive solids Solid models of basic 3D shapes that are used to create more complex solid models. (Ch. 3)



Primitive Solids

prismatic solids Solids that are formed when linear sweeps are created with a constant cross-section from end to end. (Ch. 11)

profile A closed region that consists of a group of curves and defines a boundary. (Ch. 6)

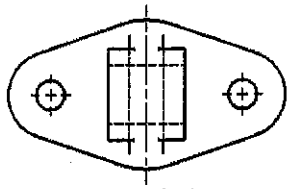


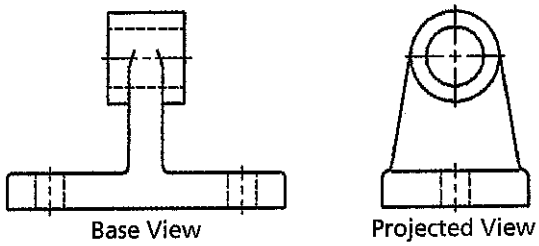
Profile

profile sketch approach The use of 2D sketched profiles as a basis for 3D solid models. (Ch. 6)

projected geometry Geometry that has been projected from its original position onto the current sketch plane for reference. (Ch. 9)

projected view A 2D view that is projected from a 3D model onto a flat surface in relation to the base view. (Ch. 8)

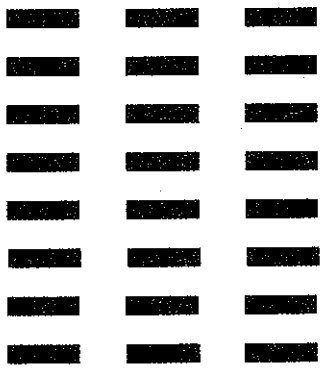




R

rapid prototyping The process of creating physical 3D prototypes directly from a computer-generated solid model. Rapid prototyping systems can build physical models from plastic resins, powdered metal, and other materials. (Ch. 8)

rectangular array An array or pattern in which a pattern leader is duplicated to create a specified number of rows and columns. (Ch. 11)

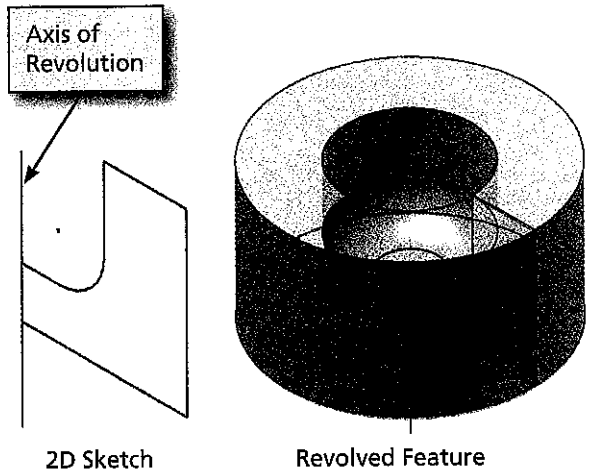


reference dimensions Dimensions added by the user. These dimensions do not control geometry; instead, they are controlled by the geometry and the feature dimensions. (Ch. 8)

reference geometry Geometry that is not used in the model but serves as a reference during model creation and during design changes. (Ch. 6)

region A closed 2D shape. (Ch. 2)

revolved feature A 3D feature that is formed by revolving a 2D sketch about a specified axis. (Ch. 10)

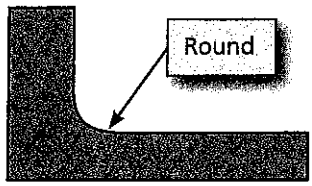


right-hand rule A rule for part rotation: If you hold the thumb, index finger, and middle finger of your right hand so that they form three right angles, the thumb points in the X direction, the index finger points in the Y direction, and the middle finger points in the Z direction. *See also:* left-hand rule. (Ch. 12)

root The final solid model that is created using various primitives and features. (Ch. 3)

rotational DOFs Rotational degrees of freedom about the various axes. There are three rotational degrees of freedom: rotation about the X axis, rotation about the Y axis, and rotation about the Z axis. (Ch. 12)

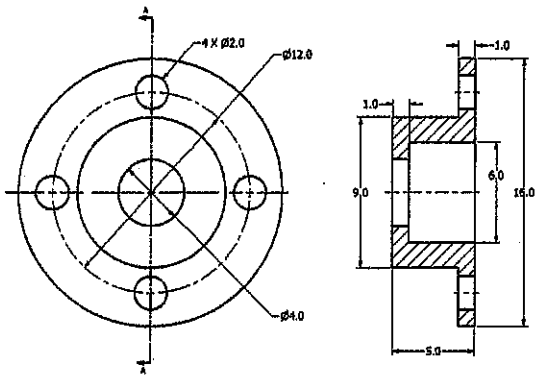
round An inside curve. *See also:* fillet. (Ch. 4)



rules The characteristics and specifications necessary to define the design intent of each construction operation. (Ch. 4)

S

section view A view of an object that shows part of the object cut away so that the viewer can see what is inside. (Ch. 10)



Front View Section View

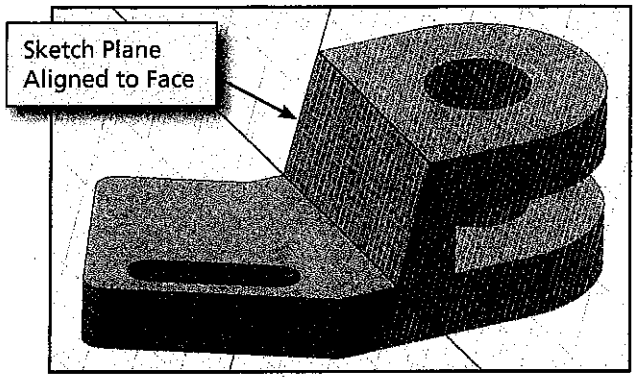
sequential record A record of the construction steps used to create a solid in order of their use. (Ch. 4)

shape-before-size design philosophy A modeling philosophy in which the basic shape or form of a new design is decided before the exact dimensions are specified. (Ch. 2)

shelling Hollowing out the inside of a solid model, leaving a shell of specified thickness. (Ch. 11)

sketched features 3D features that have been created from 2D sketches. *See also:* placed features. (Ch. 3)

sketch plane A construction tool made up of a local coordinate system. The sketch plane can be aligned to the world coordinate system, the face of an existing part, or a reference plane. (Ch. 2)

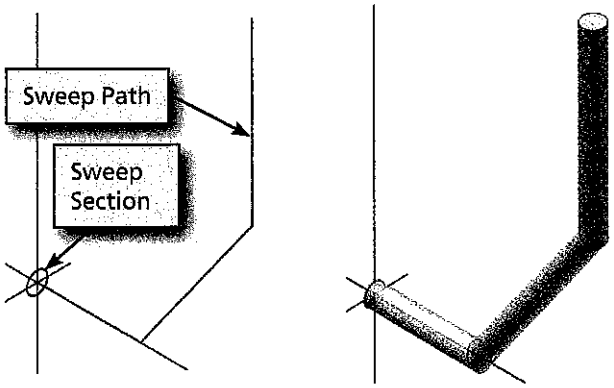


solid models CAD models that are complete, unambiguous mathematical representations of enclosed, filled volumes. CAD models have surfaces, volume, and interior structures and can be used for various types of prototypical analysis. (Ch. 1)

subassemblies Smaller assemblies that are combined to create a full assembly model. (Ch. 12)

surface models CAD models in which edges define polygonal surfaces. Surface models have surfaces, but no mass or interior structure. (Ch. 1)

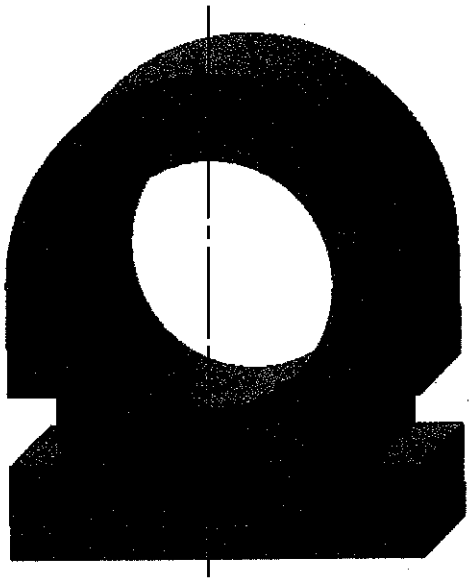
sweeping The process of moving a profile, or sweep section, through a 2D or 3D path in space to form a 3D solid object. (Ch. 11)



sweep path The path along which a profile is moved to create a 3D solid model in the sweeping technique. Also called a *trajectory*. (Ch. 11)

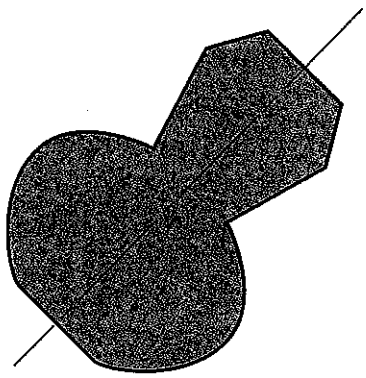
sweep section The planar profile that is swept through a specified path to create a 3D solid model using the sweeping technique. (Ch. 11)

symmetrical feature A feature in which the geometry on one side of a centerline is the exact mirror image of the geometry on the other side of the centerline. (Ch. 10)



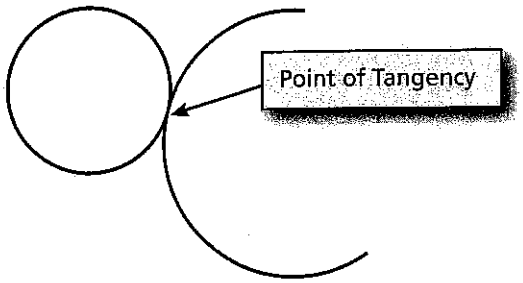
Feature with Vertical Symmetry

symmetry Having the same size and shape on both sides of a centerline. (Ch. 10)



T

tangent Arcs or circles that touch at one point only. (Ch. 5)

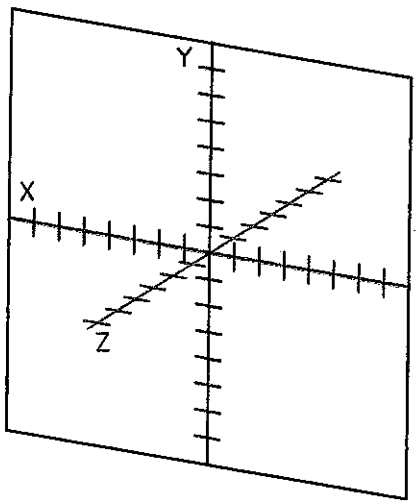


terminal branches The "leaves," or individual primitives or parts that have been combined to create a finished solid model. (Ch. 3)

termination surface The surface that signals the software to end an extrusion when the **To** option is selected for the extent. (Ch. 4)

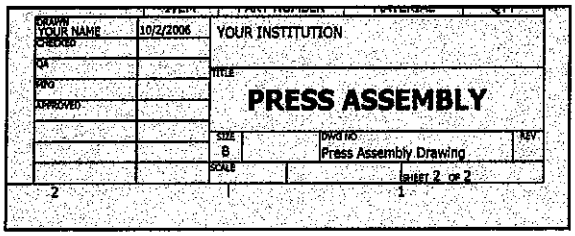
third-angle projection A method of projecting a view of a 3D object onto a 2D plane (paper) in which the object is projected from the third Cartesian quadrant. (Ch. 8)

3D Cartesian coordinate system A system of coordinates that consists of three axes at right angles to each other. The X axis runs horizontally, the Y axis runs vertically, and the Z axis runs through the intersection of the X and Y axes at 90° to both of them. (Ch. 2)



3D Cartesian Coordinate System

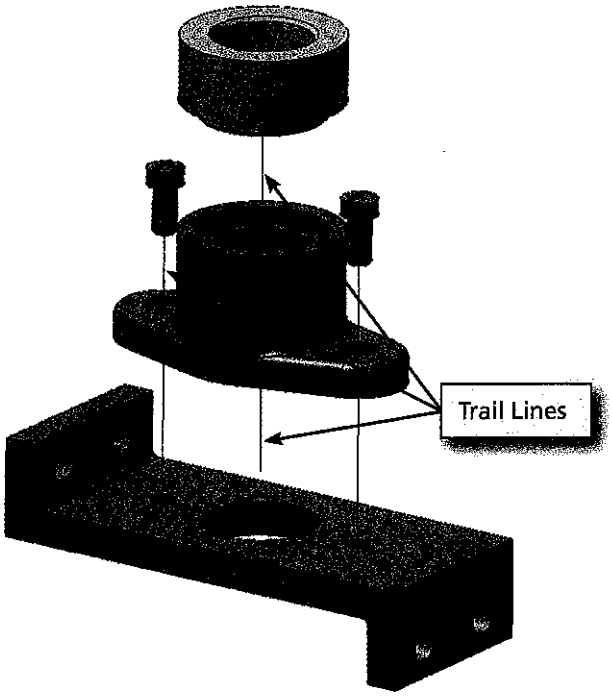
title block The area in the lower right corner of a 2D drawing that contains information about the drawing. (Ch. 3)



Title Block

top-down approach An approach to assembly modeling in which the assembly model is created first using names or symbols instead of the actual parts. The details of the individual parts are specified as the project gets further along. (Ch. 12)

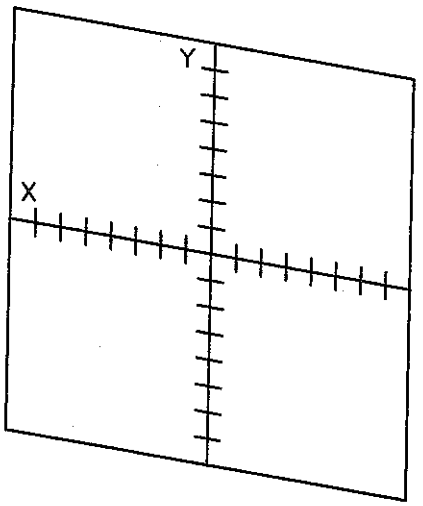
trail lines Lines from each part in an exploded assembly to show exactly where the part fits into the assembly. (Ch. 13)



trajectory The path along which a profile is moved to create a 3D solid model in the sweeping technique. Also called a *sweep path*. (Ch. 11)

translational DOFs Translational degrees of freedom along the axes of the Cartesian coordinate system. There are three translational degrees of freedom: movement along the X axis, movement along the Y axis, and movement along the Z axis. (Ch. 12)

2D Cartesian coordinate systems A system of coordinates that consists of two axes at right angles to each other. The X axis runs horizontally and the Y axis runs vertically. This coordinate system has no depth scale, so it can only be used for 2D sketches and planar objects. (Ch. 2)



2D Cartesian Coordinate System

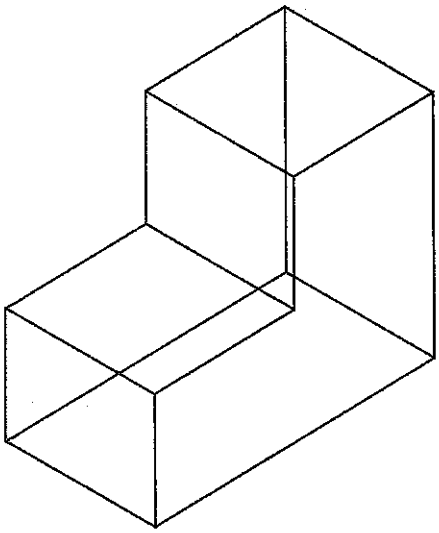
U

user coordinate systems (UCS) Coordinate systems that are defined by the user of the CAD software based on drawing needs. Like local coordinate systems, UCSs are defined within the individual electronic files. (Ch. 2)

user parameters Parameters that can be defined by the user to supplement model parameters. (Ch. 5)

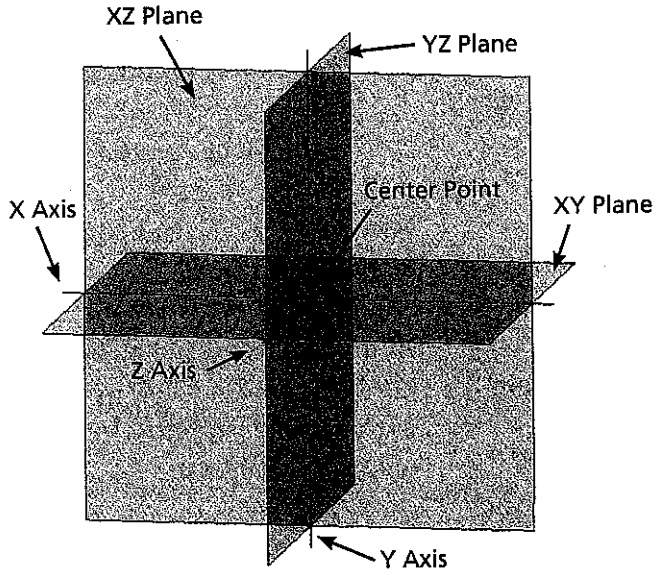
W

wireframe models CAD models consisting of points and edges, which are straight lines connecting appropriate points. Wireframe models have no surfaces and no volume. (Ch. 1)



Wireframe Model

work axes The three axes that make up the 3D Cartesian coordinate system: X, Y, and Z. These axes are available by default for use in creating features when the BORN method is used. (Ch. 7)



work features The default planes, axes, and center point in Inventor, which can be referenced during feature construction. Specifically, the work features consist of the YZ, XZ, and XY planes, the X, Y, and Z axes, and the origin of the coordinate system (center point). (Ch. 7)

work planes The three planes that make up the 3D Cartesian coordinate system: YZ, XZ, and XY. These planes are available by default for use in creating features when the BORN method is used. (Ch. 7)

world coordinate system The Cartesian coordinate system, as it is defined and used in CAD software. (Ch. 2)

X

X axis In a 2D or 3D Cartesian coordinate system, the axis that runs horizontally. (Chs. 2, 7)

XY plane The plane formed by the X and Y axes of a 2D or 3D Cartesian coordinate system. (Chs. 2, 7)

XZ plane The plane formed by the X and Z axes of the 3D Cartesian coordinate system. (Chs. 2, 7)

Y

Y axis In a 2D or 3D Cartesian coordinate system, the axis that runs vertically. (Chs. 2, 7)

YZ plane The plane formed by the Y and Z axes of the 3D Cartesian coordinate system. (Chs. 2, 7)

Z

Z axis In a 2D or 3D Cartesian coordinate system, the axis that runs at 90° angles to both the X axis and the Y axis to show depth. (Chs. 2, 7)

zoom Change to a different magnification in the graphics window. (Ch. 1)