2D plane: A flat, infinite 2D surface. (Ch. 3)

active standard: The standard that is currently in use in a model or drawing file. (Ch. 19)

aligned dimension: A dimension used to define an object or feature that is not vertical or horizontal. (Ch. 4)

alt-drag: Establishing assembly constraints, including mate, flush, tangent, and insert constraints, by dragging one component to another component; also called drag-mate. (Ch. 18)

angular dimension: A dimension used to define the angle between two lines. (Ch. 4)

arc: A circular curve in which all of the points are an equal distance from the center point. (Ch. 3)

arrowless dimensioning: A dimensioning method that provides coordinates from established datums that are usually located at the corner of the part or the axis of a feature. Also called rectangular coordinate dimensioning without dimension lines or ordinate dimensioning. (Ch. 19)

assembly: A grouping of one or more design components. (Ch. 1, 17)

assembly drawing: A 2D representation of an assembly. (Ch. 19)

assembly constraints: Constraints that establish geometric relationships and positions between one component face, edge, or axis and another component face, edge, or axis. (Ch. 18)

auxiliary view: A view used to show the true size and shape of an inclined surface that is not parallel to any of the projected views, including the front, top, bottom, left-side, right-side, and back views. (Ch. 19)

axis of rotation: The pivot point around which the selected geometry is copied. (Ch. 4)

balloon: A shape, usually circular, that is connected to an assembly component by a leader. It contains an identification number or letter that refers to an item in the parts list. (Ch. 19)

base environment: The overall working environment, within which secondary environments exist. (Ch. 1)

base feature: The initial model feature, on which all others are based. (Ch. 1, 5)

baseline dimensioning: A dimensioning method in which the size and location of features are given in reference to a datum. Also referred to as datum dimensioning. (Ch. 19)

base view: The first view placed in a drawing, to which all other views are added. (Ch. 19)

bend radius: The inside radius of a formed feature. (Ch. 15)
bend relief: Relief typically added to a sheet metal part to relieve stress, or the tear, that occurs when a portion of a piece of material is bent. (Ch. 15)
bent: Formed using a brake, die, mandrel, roller, or similar tools. (Ch. 15)
border: A rectangle or polygon near the edge of the drawing sheet that defines the usable drawing area of the drawing sheet. Borders may also include zone numbers and center marks. (Ch. 19)
boundary patch: A surface formed by patching the space within a selected closed region. (Ch. 13 supplement)
bowtie grips: Handlebar endpoints used to adjust the shape of a spline. (Ch. 3)
browser bar (browser): A panel that displays all the items in the current model or drawing. (Ch. 1)
cascading menu: A secondary menu that contains options related to the chosen menu item. (Ch. 1)
catalog feature: A feature, part, or assembly stored in a catalog that can be inserted into a part model as a feature. (Ch. 14)
centerline: A line that defines an axis of symmetry or the center of a circular feature. (Ch. 3)
center of gravity: The center of model mass, where balance occurs. (Ch. 6)
center point: The intersection point of the X, Y, and Z axes in 3D space, or 0,0,0. (Ch. 3)
chamfers: Angled planar faces added to lines or curves. (Ch. 3) Angled planar faces placed on a feature edge. (Ch. 8)
child node: Subordinate nodes that create, are associated with, or are consumed by the parent node item. (Ch. 1)
circular feature pattern: Occurrences of features copied and positioned a specified distance apart around an axis. (Ch. 10)
circular pattern: An arrangement of copies of a feature around an imaginary circle, a designated number of times, and at a specified distance apart. (Ch. 1)
circumscribed: Describes a polygon in which the flats are tangent to an imaginary circle; circumscribed polygons are measured across the polygon flats. (Ch. 3)
closed loop: A sketch that is fully closed and does not contain any gaps or openings. (Ch. 3)
coil: A spiral, or helix, feature used primarily to create springs, detailed threads, and similar items. (Ch. 5)
coincident constraint: A constraint that forces two points to share the same location. (Ch. 3)
combs: Lines added to the spline to help illustrate and analyze the spline curvature. (Ch. 3)
components: The individual parts and subassemblies used to create an assembly. (Ch. 1, 17)
composite iMates: Two or more iMates linked together and added to a single component; used for the same assembly operation. (Ch. 18)
constant fillets and rounds: Fillets and rounds that have a curve radius that does not change. (Ch. 8)
constraints: Parameters that control the size, location, and position of model elements, including sketches and features. (Ch. 1) Restrictions applied to sketches to define sketch geometry in reference to other sketch geometry. Also called geometric constraints. (Ch. 3)
construction geometry: Geometry used for construction purposes only. Inventor cannot use construction geometry to build sketched features. (Ch. 3)
consumed: Used up in the creation of a model or feature. (Ch. 5)
context-sensitive shortcut menu: Menu in which only items associated with the current work environment and application are available. (Ch. 1)
control keys: Shortcut key combinations that include the [Ctrl] key and a character key. (Ch. 1)
coordinate system: The system of XYZ coordinate values that defines the location of points in 3D space. (Ch. 3)
**corner chamfers**: Angled faces that replace square corners on sheet metal features. (Ch. 16)

**corner relief**: Relief typically added to a sheet metal part to relieve stress at a bend corner at the intersection of two or three faces. (Ch. 15)

**corner rip**: A feature that opens closed, usually square, corners. (Ch. 16)

**corner round**: A curve placed at an inside or outside sheet metal corner. (Ch. 16)

**corner seams**: Features that add or remove material to form a gap at sheet metal part corners. Corner seams create an appropriate corner transition for folding and to allow for unfolding. (Ch. 16)

**counterbored hole**: A drilled hole that has a larger-diameter cylindrical opening at the top; typically used when a flush surface is necessary, such as to hide a binding screw head. (Ch. 7)

**countersunk hole**: Similar to a counterbored hole, but the recess is tapered, resulting in a conical shape that is often used to hide a screw head. (Ch. 7)

**curve**: A straight or bent continuous object, such as a line, arc, spline, or circle. (Ch. 3)

**cut**: Remove volume from an existing extrusion by subtracting a new extrusion from it. (Ch. 5) Any process, such as shearing, punching, or laser, water jet, or similar process, used to remove material. (Ch. 15)

**cutting-plane line**: A line that represents the cutting plane of the section, which is the location where the view is sliced to show interior features. (Ch. 19)

**cutting tool**: A surface, quilt, 2D sketch curve, work plane or existing feature face intersecting the surface to trim that provides an edge to which the item is trimmed. (Ch. 13 supplement)

**dangling geometry**: A condition that results when additional positioning information is required in order for iFeature insertion to occur; primarily due to issues with the initial iFeature sketch and existing feature geometry. (Ch. 16)

**database**: A system that stores every model characteristic, including calculations, sketches, features, dimensions, geometric constraints, when each piece of the model was created, and all other model parameters and properties. (Ch. 1)

**datum**: A theoretically exact point, axis, or plane from which the location or geometric characteristics of features originate. (Ch. 16)

**datum dimensioning**: A dimensioning method in which the size and location of features are given in reference to a datum. Also referred to as baseline dimensioning. (Ch. 19)

**decals**: Images applied to a part or assembly to display information or decorate a product. (Ch. 13)

**demote**: Group more than one part in an assembly to create a subassembly. (Ch. 17)

**dependents**: Assembly component files referenced by the assembly. (Ch. 17)

**dependent views**: Views projected from and linked to another view, such as a base view. (Ch. 19)

**derived components**: Features that can contain a complete model consisting of several features, or even multiple parts; often used as a base feature. (Ch. 1) A saved part or assembly that can be inserted in a part as a feature. (Ch. 14)

**design session**: Time spent working on a project, including analyzing design parameters and using Inventor. (Ch. 2)

**detail view**: A view that shows a small, complex part feature at a larger scale. (Ch. 19)

**dialog box**: A window-like part of the user interface that contains various kinds of information and settings. (Ch. 1)

**diameter**: The distance across a circle from one side to the other through the center. (Ch. 4)

**diameter dimension**: A dimension used to define the diameter of a circle or circular object. (Ch. 4)
**dimension:** A measurement that numerically defines the size and location of sketch geometry, such as the length of a line, diameter of a circle, or radius of an arc. (Ch. 4) Specifications of the size and shape of object features so that parts can be manufactured; along with notes and other text, also specify the location and characteristics of geometry and surface texture. (Ch. 19)

**docked:** Describes interface items that are locked into position on an edge of the Inventor window (top, bottom, left, or right). (Ch. 1)

**document units:** The units used to define the linear, angular, time, and mass measurements and precision in models and drawings. (Ch. 2)

**double bend:** A bend between two parallel faces that are not coplanar. (Ch. 15)

**drag-mate:** Establishing assembly constraints, including mate, flush, tangent, and insert constraints, by dragging one component to another component; also called *alt-drag*. (Ch. 18)

**drawing annotation tools:** Tools that allow you to create annotations such as dimensions, notes, and other text on drawings. (Ch. 19)

**drawing dimensions:** Dimensions added to the drawing using Inventor’s drawing annotation tools. (Ch. 19)

**drawing sheet:** A representation of the physical limits of the paper size on which the drawing will be printed. (Ch. 19)

**drawings:** 2D representations of models containing views, dimensions, and annotations. (Ch. 1)

**drilled hole:** The most basic hole type, with no counterbore, spotface, or countersink where the hole begins. (Ch. 7)

**driven:** Manipulated to see the amount of movement between components, pause movement, see adaptivity, and detect collisions between components. (Ch. 18)

**driven dimension:** A dimension used for reference purposes only. Reference dimensions are enclosed in parentheses to show that they are driven. (Ch. 4)

**ellipse:** An oval-like shape that contains both a major axis and a minor axis. (Ch. 3)

**embossing:** The process of raising shapes or text off the surface of an object that has volume, such as a block; the opposite of engraving. (Ch. 13)

**engraving:** The process of cutting into, or impressing, shapes or text into the surface of an object that has volume; the opposite of embossing. (Ch. 13)

**external threads:** Thread forms on an external feature such as a pin, shaft, bolt, or screw. (Ch. 9)

**extrusion:** A surface or solid that has a fixed cross-sectional profile determined by a sketch profile. The sketch profile is extended (extruded) along a linear path to create the 3D feature or part. (Ch. 5)

**face draft:** A taper placed on a part surface. (Ch. 9)

**feature pattern:** An arrangement of copied existing features, generating occurrences of the features. (Ch. 1) An arrangement of features in a specific pattern, or configuration; created using feature pattern tools. (Ch. 10)

**fillets:** Rounded interior corners; fillets add material to corners. (Ch. 3) A curve placed at the inside intersection of two or more faces, adding material to a feature. (Ch. 8)

**flat angle:** The number of degrees a coil end travels without pitch. (Ch. 5)

**flat end:** A type of coil end in which the first or last coil is adjusted to create a flat start or finish for the spring. (Ch. 5)

**flat pattern:** A 2D drawing representing the final, unfolded part. (Ch. 15)

**floating:** Describes interface items, displayed within a border, that can be freely resized or moved. (Ch. 1)

**flush solution:** A constraint that positions two faces along the same plane, facing the same direction. (Ch. 18)
**flyout**: A button that presents additional, related tool buttons, much like a cascading menu. (Ch. 1)

**fly-through**: A viewing process that shows how it would look if you could fly in and around the actual product you are modeling. (Ch. 6)

**frequently used subfolder**: A virtual folder within a project that stores the paths to folders and files you use frequently. (Ch. 2)

**fully constrained model**: A model that has no freedom of movement. (Ch. 1)

**full radius fillets and rounds**: Fillets and rounds controlled by the linear dimension of a feature, such as the thickness of a part or width of a slot, producing half of a circle or cylinder; most often associated with a round. (Ch. 8)

**general notes**: Notes that apply to the entire drawing. General notes are usually placed together in the lower-left or upper-right corner of the drawing or in the title block. (Ch. 19)

**geometric constraints**: Restrictions applied to sketches to define sketch geometry in reference to other sketch geometry. Also called *constraints*. (Ch. 3)

**geometric dimensioning and tolerancing (GD&T)**: The dimensioning and tolerancing of individual features of a part where the permissible variations relate to characteristics of form, profile, orientation, runout, or the location of features. (Ch. 19)

**grab bars**: Two thin bars at the top or left edge of a docked or floating item; used to move the item. (Ch. 1)

**graphical user interface (GUI)**: On-screen interface items. (Ch. 1)

**grounded component**: An assembly component that is fixed in position, has no freedom of movement, and cannot be driven. (Ch. 17)

**grounded work point**: A work point completely fixed to an X, Y, Z coordinate at which it is placed. (Ch. 11)

**guide rail**: A 2D or 3D sketched curve that is used with the sweep path to manipulate and further control the shape of a sweep. (Ch. 12)

**guide surface**: A surface that helps control the shape of a sweep along a complex path. (Ch. 12)

**height**: In a coil, the total depth of the coil from the center of the starting profile to the center of the ending profile. (Ch. 5)

**help string**: A short description of what happens if you select a tool or option over which the cursor is hovering, or if a tool is selected, a prompt indicating the appropriate action is shown. (Ch. 1)

**hem**: Flanges used to add strength to or relieve the sharpness of exposed edges, or to connect separate edges or parts together. (Ch. 16)

**hot keys**: Single character keys on the keyboard that allow you to access certain predefined tools. (Ch. 1)

**icon**: A small graphic representing an application, file, or tool. (Ch. 1)

**i-drop**: The process of dragging and dropping shared content into component files, or the tool used for this process. (Ch. 17)

**iFeature**: An existing feature or set of features you create and then save and store in a catalog to be used in other models. (Ch. 1) A stored feature that can be inserted in a part as a feature. (Ch. 14)

**iMates**: Constraints placed on an individual component that are later used for assembly. (Ch. 18)

**included angle**: The angle between two selected edges, curves, axes, faces, planes, or a combination of objects, such as an edge and a face. (Ch. 6)

**included file**: A separate project file linked to the current project. (Ch. 2)

**increment**: A set amount by which values increase in equal steps. For example, with an increment of 2, a size would increase to 4, 6, 8, 10, and so on. (Ch. 14)
inferred: Automatically detected using logic. (Ch. 3)
inscribed: Describes a polygon in which the corners touch an imaginary circle; inscribed polygons are measured from the corners. (Ch. 3)
interface: The tools and techniques used to provide information to and receive information from a computer application. Also called a user interface. (Ch. 1)
internal threads: Thread forms on an internal hole feature. (Ch. 9)
iProperties: Inventor file properties used to define a variety of file and design characteristics. (Ch. 2)
isometric view: A 3D view in which all three axes are shown at equal angles ($120^\circ$) with the plane of projection. (Ch. 6)

J
join: Combine two or more existing features to create a single feature. (Ch. 5)

K
k-factor: A multiple, typically between .25 and .5, that locates the neutral axis. (Ch. 15)

L
leader: A line that connects the beginning or end of a note to the feature it describes. Leaders usually have a horizontal shoulder on the end nearest the text. The other end has an arrow pointing to the feature. (Ch. 19)
left-hand threads: Threads that move a left-hand threaded bolt forward in a counterclockwise direction. (Ch. 7)
library: A folder that contains files used in a project or several different projects. (Ch. 2)
library search paths: The locations in which Inventor looks for library files on the computer's hard drive or on the network. (Ch. 2)
linear dimension: A type of dimension used to define the vertical and horizontal size and location of object features. (Ch. 4)
local notes: Notes that apply to a specific feature or features on the drawing. Also called specific notes. (Ch. 19)

loft: A feature that references and blends two or more sections located on different planes. (Ch. 12)
loft centerline: A rail that acts as a path for blending sections along and symmetrically around the centerline sketch. (Ch. 12)
lump: Any set of external feature or surface faces created when you develop a solid model. (Ch. 13 supplement)

M
mate solution: A constraint that places two faces along the same plane facing in opposite directions, two axes collinear to each other, two edges collinear to each other, or two points matched together. (Ch. 18)
mirrored feature: mirrored features: A mirror image of an existing feature created symmetrically over a specified plane. (Ch. 1, 10)
mirror plane: A plane of symmetry about which features are mirrored. (Ch. 10)
miter gap: Space between faces created during a corner seam or miter operation. (Ch. 15)
model dimensions: Dimensions that were used to create and constrain the model from which drawing content has been extracted. (Ch. 19)
modeling failure: The result of conflicting constraints that are impossible to apply to the model. (Ch. 1)
model parameters: Parameters that relate to the model. Model parameters are added when you insert a model view or add model information, such as dimensions. (Ch. 3)
model space: A space, or environment, in which the model defines the display orientation, regardless of the position of the model in the graphics window; the center is associated with the model pivot point. (Ch. 6)
monodetail drawing: A drawing of a single part on one sheet. (Ch. 19)
motion constraints: Assembly constraints that identify how movable components should move in reference to other movable components, using a specified ratio and direction. (Ch. 18)
**multidetail drawing**: A drawing of several parts on one sheet. (Ch. 19)

**multiple document interface**: An interface that allows you to have several documents or document views open at the same time. Also called *multiple design interface*. (Ch. 1)

**natural end**: A type of coil end that occurs as the natural result of the pitch, revolution, height, and profile of the coil. (Ch. 5)

**network**: Several ribs or webs created using the same direction and thickness. (Ch. 7)

**neutral axis**: The axis of a bend radius where neither stretching nor compressing occurs. (Ch. 15)

**nominal size**: The designated size of a commercial product. (Ch. 7)

**nominal value**: The value of a commercial product; intended to be the true drawn size without any specified limits. (Ch. 4)

**oblique view**: A 3D view in which the plane of projection is parallel to the front surface, and a receding angle is applied. (Ch. 6)

**offset**: Form objects parallel to the specified geometry at a specified distance apart. (Ch. 4) When referring to the Thicken/Offset tool, the process of offsetting a surface from a face or surface, similar to offsetting a work plane from a face. (Ch. 9) When referring to threads, the distance from the edge of the face to the beginning of threads. (Ch. 9)

**open loop**: A sketch that includes a gap(s) between objects. (Ch. 3)

**open sketch profile**: A sketch profile that does not form a closed loop. (Ch. 5)

**ordinate dimensioning**: A dimensioning method that provides coordinates from established datums that are usually located at the corner of the part or the axis of a feature. Also called *rectangular coordinate dimensioning without dimension lines* or *arrowless dimensioning*. (Ch. 19)

**origin**: The center point (0,0,0) of the model's XYZ coordinate system. (Ch. 3)

**orphaned annotations**: Annotations that have been moved away from a drawing view associated with model geometry. (Ch. 19)

**orthographic view**: A 2D view, or projection, in which the line of sight is perpendicular to a surface, such as the front of an object or the XY plane. (Ch. 6)

**over-constrained model**: A model with too many constraints. (Ch. 1)

**pan**: Reposition the display of objects in the graphics window. (Ch. 3)

**panel bar**: A panel-like window that appears by default on the left side of the Inventor graphics window. Panel bars are the primary default location for accessing design tools. (Ch. 1)

**parallel**: A geometric construction that specifies that objects such as lines and ellipse axes will never intersect, no matter how long they become. (Ch. 3)

**parameters**: Characteristics that control the size, shape, and position of model geometry. (Ch. 1) Shape and size limits placed on sketches and features. (Ch. 4)

**parametric solid modeling**: A form of modeling in which parameters and constraints drive the model form and function to produce models that contain object volume and mass data that can be used to analyze internal and external object characteristics. (Ch. 1)

**parent node**: An item in the tree structure, similar to a folder, that is associated with subordinate child nodes. (Ch. 1)

**part**: An item or product or an element of an assembly. (Ch. 1)

**partial auxiliary view**: An auxiliary view that shows the true size and shape of only the inclined surface, eliminating any projected geometry that may be foreshortened. (Ch. 19)

**parts list**: A table that records and displays the parts and subassemblies used to create an assembly. (Ch. 19)

**path**: A guide, or route, for creating sketched features. (Ch. 3)
pattern occurrences: Representations of patterned features that identify how many features are present because of the pattern operation. (Ch. 10)

perpendicular: A geometric construction that defines a 90° angle between objects such as lines and ellipse axes. (Ch. 3)

pitch: The distance parallel to the axis between a point on one coil spiral to the corresponding point on the next coil spiral. (Ch. 5) The distance parallel to the axis from a point on one thread to the corresponding point on the next thread. (Ch. 7)

pivot point: The point that acts as the center point when you are viewing and rotating model space objects. (Ch. 6)

placed features: Features added to an existing feature without using a sketch. (Ch. 1, 5, 8)

placed sections: Loft sections that are created without a sketch and are placed along a selected centerline. Placed sections are calculated based on the loft cross section at the selected location. (Ch. 12)

profile: The side or section outline of a sketched feature. (Ch. 3)

projects: Files that manage and organize folders and files for specific design jobs. (Ch. 2)

promote: Add to the part environment. (Ch. 13 supplement) Remove parts from a subassembly and make them individual parts in the parent assembly. (Ch. 17)

pull direction: The direction in which the casting mold is pulled or removed from the part. (Ch. 9)

pull-down menus: A text-based menu input system in which menu items appear when you pick the menu name. (Ch. 1)

punch: A press or similar tool used to form a specific shape or hole in sheet metal. Also called a sheet metal punch. (Ch. 16)

R

radius: The distance from the center of a circle or arc to its circumference. (Ch. 4)

radius dimension: A dimension used to define the radius of an arc or circular feature. (Ch. 4)

rail: A 2D or 3D sketched curve that is used in conjunction with sections to manipulate and further control the loft shape. (Ch. 12)

read-only: A file open option that allows you to view a file, but not make changes to it. (Ch. 2)

realtime zooming: Zooming that can be viewed as it is performed. (Ch. 6)

rectangular coordinate dimensioning without dimension lines: A dimensioning method that provides coordinates from established datums that are usually located at the corner of the part or the axis of a feature. Also called ordinate dimensioning or arrowless dimensioning. (Ch. 19)

rectangular feature pattern: Occurrences of features copied and positioned a specified distance apart, in rows and columns. (Ch. 10)

rectangular pattern: An arrangement of copies of a feature into a designated number of rows and columns placed a specified distance apart. (Ch. 17)

regular polygon: A geometric shape with three or more sides, such as a triangle, square, or hexagon, with all sides being equal in length and symmetrical about a common center. (Ch. 3)

revision table: A table that records drawing changes; usually placed in the upper-right corner of the drawing. Also called a revision history block or revision block. (Ch. 19)

revision tag: A symbol that identifies the location at which the engineering change occurs. The tag corresponds to a specific entry in the revision table. Also called a revision symbol. (Ch. 19)

revolution: A feature created in a circular path around an axis; also called a revolved feature. (Ch. 5) In a coil, one complete spiral, or 360° loop. (Ch. 5)

Q

quilt: A set of combined surfaces. (Ch. 13 supplement)
**revolved feature**: A feature created in a circular path around an axis. Also known as a *revolution*. (Ch. 5)

**rib**: A closed section of material usually added to reinforce a part without adding excessive material or weight. (Ch. 7)

**right-hand threads**: Threads that move a right-hand threaded bolt forward in a clockwise direction. (Ch. 7)

**rounds**: Rounded exterior corners; rounds remove material from corners. (Ch. 3) A curve placed on the exterior intersection of two or more faces, removing material from a feature. (Ch. 8)

**scale factor**: The amount of enlargement or reduction. (Ch. 4)

**screen space**: A space, or environment, in which the graphics window controls model display; the center is located at the center of the graphics window. (Ch. 6)

**sculpt**: The process of using intersecting surfaces to add or remove solid mass. (Ch. 13 supplement)

**sections**: Sketches and existing feature faces used to develop loft features. (Ch. 12) A view that splits a part along a cutting-plane line to expose the interior features of the part. Also called a *section view*. (Ch. 19)

**section view**: A view that splits a part along a cutting-plane line to expose the interior features of the part. Also called a *section*. (Ch. 19)

**setback**: Point at which a fillet or round on one edge begins to combine with a fillet or round of at least two other edges. (Ch. 8)

**shared content**: Files available on the Internet, such as bolts from a bolt manufacturer, or components accessible on an intranet system, such as standard parts that are used for developing assemblies. Also called *third-party content*. (Ch. 17)

**sharing**: Making a sketch available for additional features after it has been used to create a feature. (Ch. 5)

**sheet formats**: Predefined, multiview drawing sheet templates that contain a default border and title block for various standard sheet sizes. (Ch. 19)

**sheet metal punch**: A press or similar tool used to form a specific shape or hole in sheet metal. Also called a *punch*. (Ch. 16)

**shell**: An operation that removes material from a feature and creates a hollow space or opening. (Ch. 9)

**shortcut keys**: Keyboard key combinations that allow you to access predefined tools. (Ch. 1)

**shortcut menus**: Menus that allow access to tools and options by right-clicking anywhere in the graphics window or on an object or selection. (Ch. 1)

**sketch**: A 2D drawing that provides the profile and/or guide for developing a sketched feature. (Ch. 1)

**sketch center points**: Points used to define the location of center points for features that reference center points, such as holes and sheet metal punches. (Ch. 3)

**sketched features**: Features such as extrusions, revolutions, sweeps, lofts, and coils that are built from a sketch. (Ch. 1, 5)

**sketch helix**: A winding spiral shape primarily used to create springs, detailed threads, and similar items. (Ch. 12)

**sketch pattern**: Multiple arranged copies, or a pattern, of sketch shapes. (Ch. 4)

**sketch points**: Points used for construction purposes to help you develop sketch geometry. (Ch. 3)

**spacing**: In patterning, the distance between occurrences based on the width of the selected features and the distance between the copies. (Ch. 10)

**specific notes**: Notes that apply to a specific feature or features on the drawing. Also called *local notes*. (Ch. 19)

**spline**: A complex curve defined by control points along the curve. (Ch. 3)

**split**: A feature that removes a portion of a model or divides faces at a separation sketch or plane. (Ch. 13)

**spotface**: Similar to a counterbore, but shallower; typically applied when a flush surface is necessary, such as to hide a flat washer, or in casting applications. (Ch. 7)
**standard**: A set of styles and other general drawing preferences that has been agreed upon and recommended for use by an industry, government, military, or standards-setting organization. (Ch. 19)

**steering wheels**: Circular navigation tools that allow you to navigate around a model. (Ch. 6)

**stitched**: Two or more surfaces combined to form a single surface or quilt. (Ch. 13 supplement)

**style library**: A folder, Design Data by default, that houses styles in XML file format. (Ch. 6, 15)

**subassembly**: An assembly placed in a larger assembly, such as switch, or spring assembly; subassemblies may be used more than once in the final assembled product. (Ch. 1, 17)

**surface extrusion**: A volumeless shape that is primarily used for construction purposes, allowing you to generate advanced models. (Ch. 5)

**surface finish**: The allowable roughness, waviness, lay, and flaws on a surface. (Ch. 19)

**sweep**: A feature created by guiding, or sweeping, a sketch profile along a sketch path. (Ch. 12)

**table-driven iFeature**: An iFeature that allows you to create multiple variations of the original iFeature using information stored in a spreadsheet. (Ch. 14)

**tabular dimensioning**: A type of arrowless dimensioning in which coordinate dimensions and size dimensions are given in a table that correlates with features on the drawing with a hole tag. (Ch. 19)

**tangent constraint**: A geometric construction that specifies how a curve touches another curve at the point of tangency. (Ch. 3)

**tap**: Use a machine tool to form an interior thread. (Ch. 7)

**tapered threads**: Threads often used for pipe fittings when a liquid or airtight seal is required. (Ch. 7)

**templates**: Files with predefined settings used to begin new documents. (Ch. 1)

**thickening**: The process of adding a solid to a face or surface, similar to a solid extrusion. (Ch. 9)

**third-party content**: Files available on the Internet, such as bolts from a bolt manufacturer, or components accessible on an intranet system, such as standard parts that are used for developing assemblies. Also called shared content. (Ch. 17)

**thread class**: The designated amount, or grade, of tolerance specified for the thread, ranging from fine to coarse threads. (Ch. 7)

**threads**: Grooves cut in a spiral fashion in or around the face of a cylindrical or conical feature. (Ch. 7)

**title block**: An area on the drawing sheet that contains information about the model, company, drafter, tolerances, and other design information. (Ch. 19)

**tolerance stack**: Text that is stacked horizontally without a fraction bar. (Ch. 3)

**tool buttons**: Buttons in a toolbar, each with a specific icon, that activate a tool or option. (Ch. 1)

**tooltip**: A small text box that displays when you hover over a button, giving information about the function of the button. (Ch. 1)

**trails**: Connection graphics between components that show their relative positions in the assembly. (Ch. 1)

**transitional constraints**: Constraints that identify relationships between the transitioning path of a fixed component and a component moving along the path. (Ch. 18)

**transition angle**: The number of degrees a coil end travels, or transitions, with pitch. (Ch. 5)

**tweaks**: Component modifications made during the preparation of a presentation. (Ch. 1)

**under-constrained model**: A model with elements that are unclear, can be changed or moved, or remain undefined. (Ch. 1)
user interface: The tools and techniques used to provide information to and receive information from a computer application. Also called an interface. (Ch. 1)

user parameters: Additional parameters defined by the user. (Ch. 3)

V

variable fillets and rounds: Fillets and rounds that have different curved radii placed at precise points between the start and end of a feature edge. (Ch. 8)

vertex: When referring to filet and round setbacks, the intersection of three or more edges. (Ch. 8)

virtual component: An assembly component used primarily to define a separate bill of materials item, without creating a model. (Ch. 17)

void: Any set of internal feature faces that define a hollow area in a solid. (Ch. 13 supplement)

work axis: An axis used to create construction lines and axes. (Ch. 1) A parametric reference line that can be located anywhere in space. (Ch. 11)

work features: Features that direct the location and arrangement of other features. (Ch. 1) Construction points, lines, and surfaces that create reference elements anywhere in space to help position and generate additional features. (Ch. 11)

work planes: Planes that are used to create construction planes. (Ch. 1) Flat reference surfaces that can be located anywhere in space. (Ch. 11)

work points: Points used to create construction points. (Ch. 1) Parametric reference points that can be located on any part feature or in 3D space. (Ch. 11)

workspace: The default folder where files are located in a project. (Ch. 2)

Z

zoom in: Increase the displayed size of objects in the graphics window to view a smaller portion of the model, but in greater detail. (Ch. 3)

zoom out: Reduce the displayed size of objects in the graphics window to display more of the model, but in view less detail. (Ch. 3)