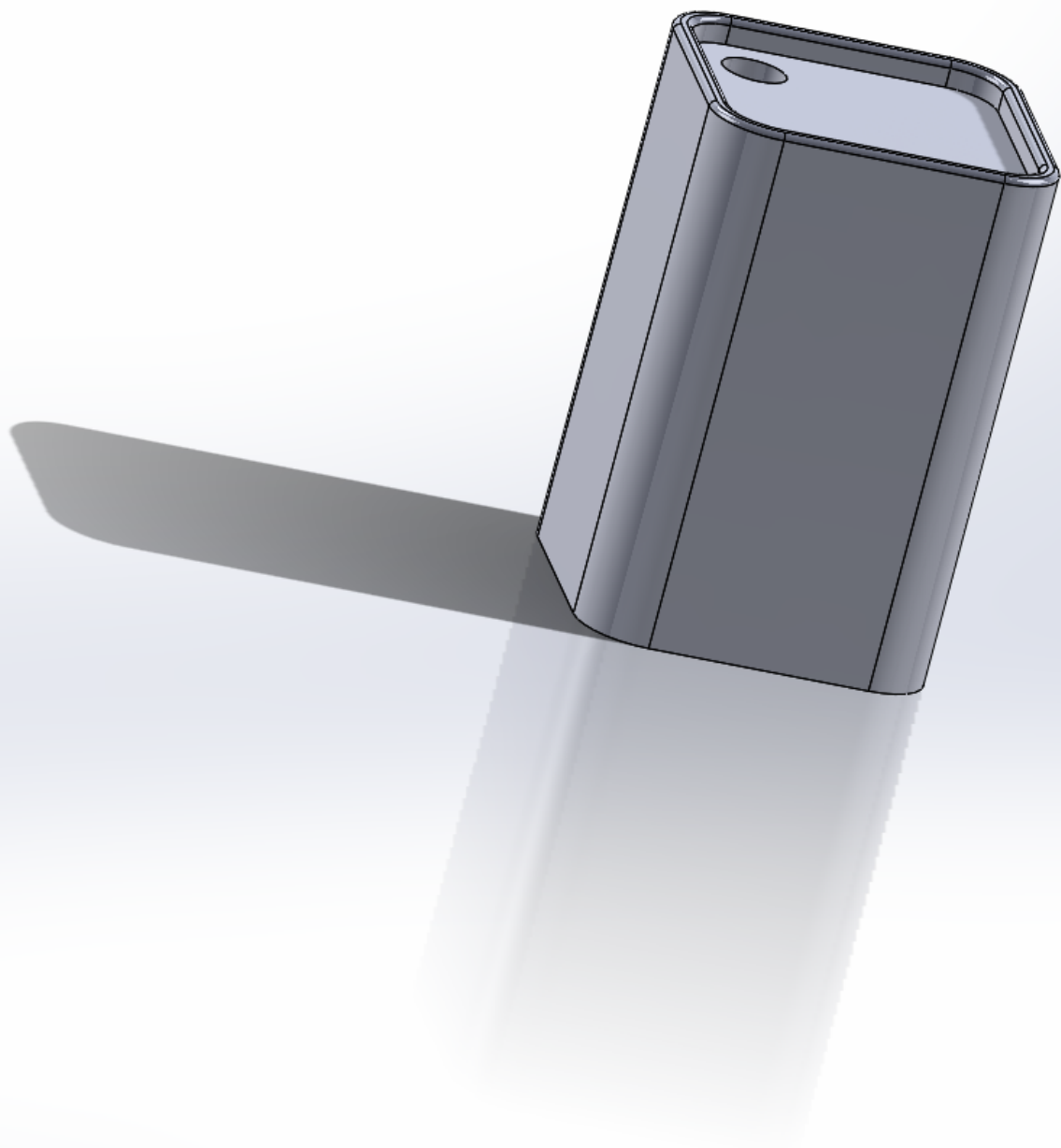
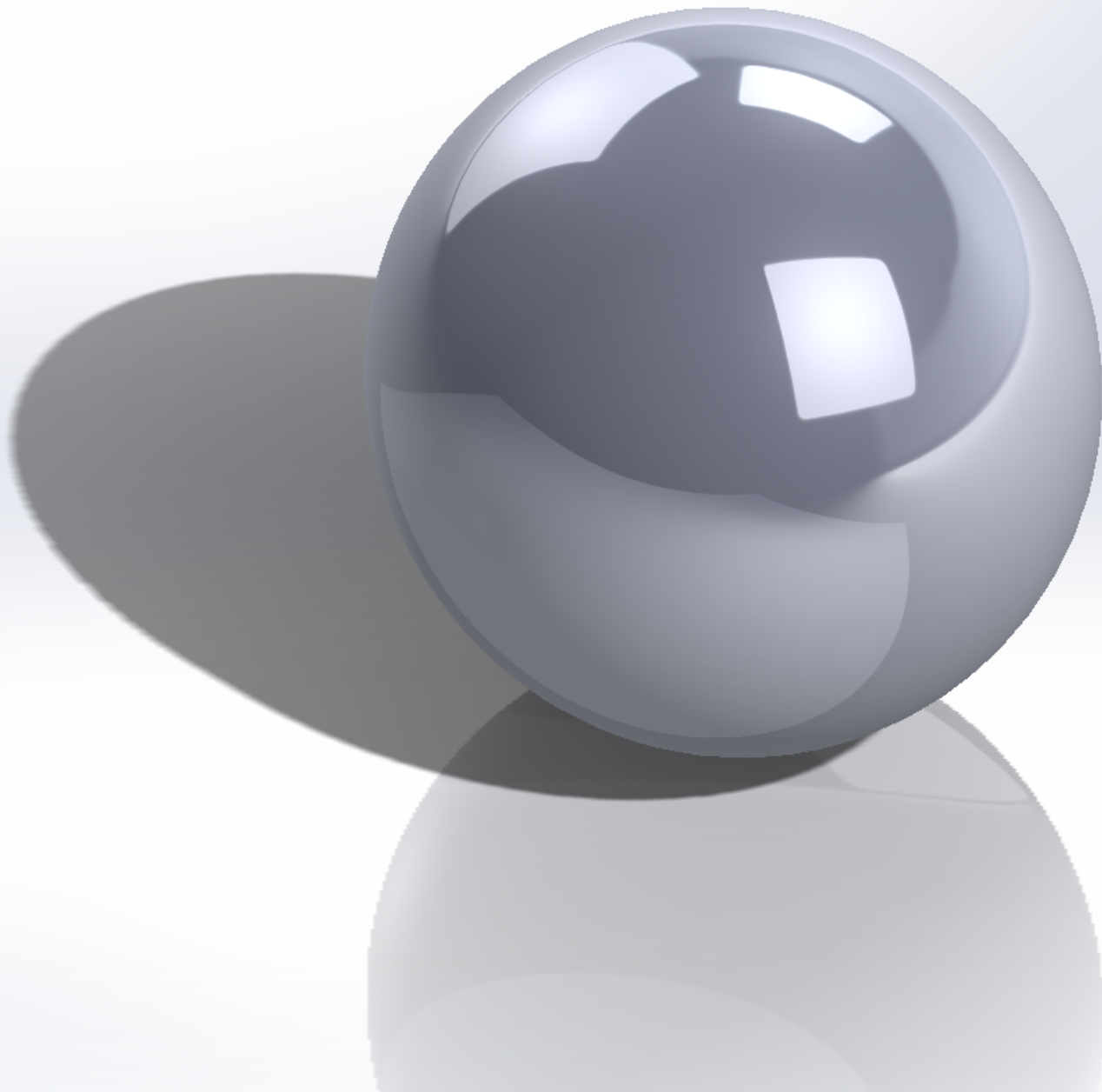


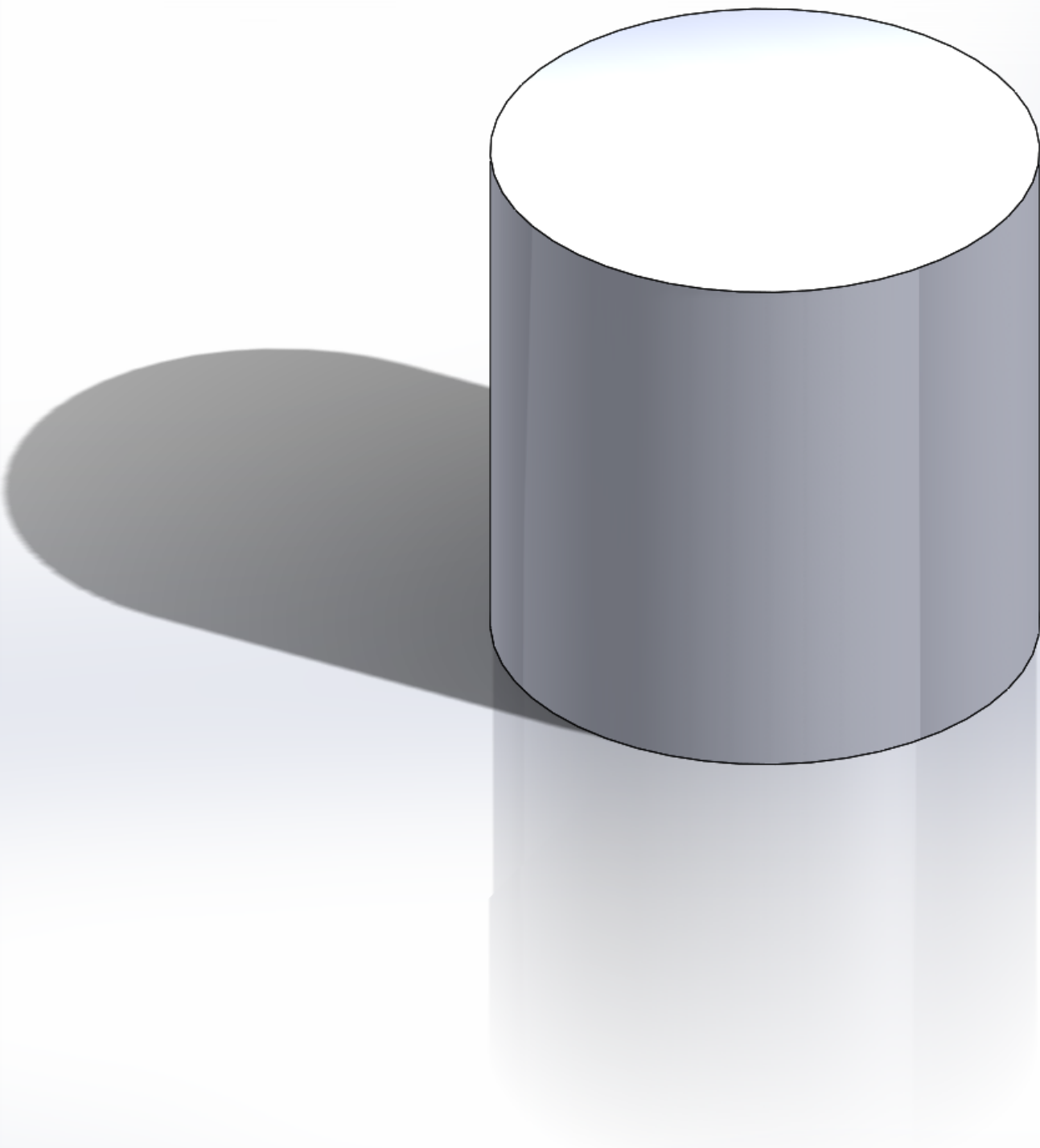
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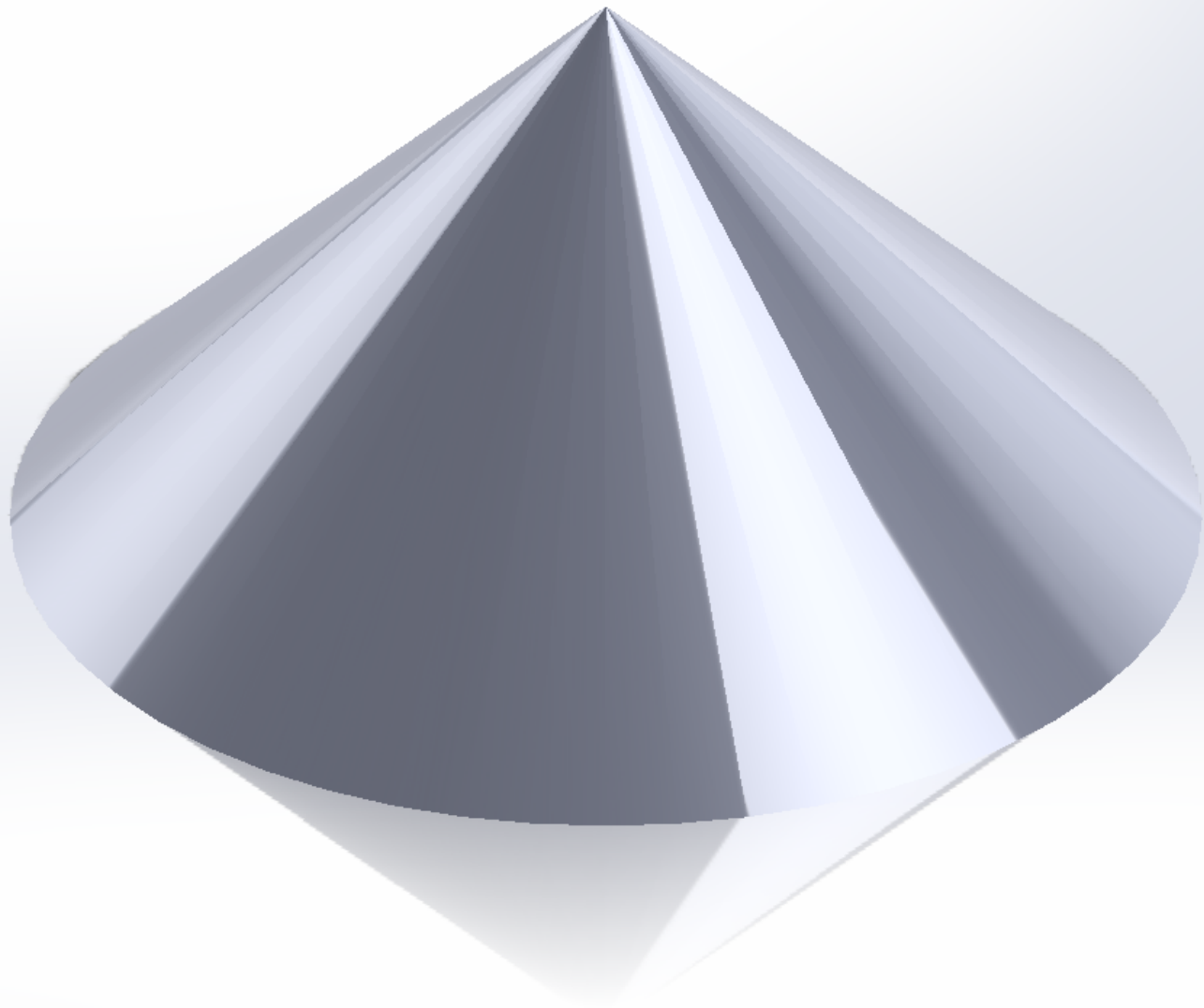
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# COLLEGEWIDE COURSE OUTLINE OF RECORD

## ENGR 116, GEOMETRIC MODELING FOR VISUALIZATION

COURSE TITLE: Geometric Modeling for Visualization

COURSE NUMBER: ENGR 116

PREREQUISITES: Demonstrated competency through appropriate assessment or earning a grade of “C” or better in MATH 100 Intermediate Algebra.

SCHOOL: Applied Science and Engineering Technology

PROGRAM: Pre-Engineering

CREDIT HOURS: 2

CONTACT HOURS: Lecture: 1 Lab: 2

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## Chapter-4 Questions

1.(A) Extruded Boss/ Base: A feature that adds materials utilizing a 2D sketch profile and a depth perpendicular to the sketch plane.

The Base feature is the first feature in the part.

(B) Fillet: A feature that round sharp edges or faces by a specified radius.

(C) Chamfer: The tool creates a beveled feature on selected edges, faces, or a vertex.

(D) Extruded Cut: A feature that removes material utilizing a 2D sketch profile and a depth perpendicular to the sketch plane.

2.(A) A Template is the foundation of a Solid Work document. A Part Template contains the Document properties such as: Dimensioning started, Units, Grid/Snap, precision, line style and Note Font.

(B) A Part is a single 3D object that consists of various feature. The filename extension for a solid works part is SLDPRT.

3. (A) click start on the windows taskbar.

(B) click ALL programs.

(C) click sold work 2014 application. The solid work program window opens.

Or If available, double-click the solid work icon on the windows Desktop to start a solid works session.

4. When you create a new part or assembly, the three default planes (Front, Right and Top) are align with specific views. The plane you select for the Base sketch determines the orientation of the part, the front drawing views and the assembly.

5. Template.

6. Front Plane, Top plane and Right Plane.

7. The Base feature is the first feature in the part.(sketch plane).

8.(A) Extruded Boss/ Base: A feature that adds materials utilizing a 2D sketch profile and a depth perpendicular to the sketch plane.

The Base feature is the first feature in the part.

(B) Extruded Cut: A feature that removes material utilizing a 2D sketch profile and a depth perpendicular to the sketch plane.

(C) Fillet: A feature that round sharp edges or faces by a specified radius.

9. Over.

10. Inadequate.

11. Under.

12. True.

13. a relation is a geometric constraint between sketch entities or between a sketch entity and plane, axis, edge or vertex.

14. Angular dimension is a style of dimensioning in which text is placed parallel to the dimension line. The aligned method of

dimensioning is not approved by the current ANSI standards but may be seen on older drawings.

15. Draft angle is the degree of taper applied to a face. Draft angle are usually applied to molds or castings.

16. An Arc requires 3 points.

17. A Multi-body part has separate solid bodies within the same part document. A WRENCH consists of two cylindrical bodies. Each extrusion is a separate body. The oval profile is sketched on the right plane and extruded with the UP to BODY option.

The BATTERY consisted of a solid body with one sketched profile.

The BATTERY is a single body part.

18. (A) Extruded Boss/Base. (B) Linear Pattern. (C) Hole Wizard. (D) Extruded Cut. (E) Revolved Boss/Base. (F) Mirror.

19. (A) Line. (B) Smart Dimension. (C) Centerline. (D) Circle. (E) Corner Rectangle. (F) 3 Point Corner Rectangle. (G) Tangent Arc. (H) Convert Entities. (I) Point. (J) Polygon.

## Chapter-5. Page 5-35 Questions Answer.

1. (A). The Revolved Base feature utilized a sketched profile on the Right plane and a centerline.

(B). The Revolved Boss feature utilized a spline sketched profile.

(C). Revolved Cut feature used to remove material by revolutions.

A Revolved Cut requires a centerline, a sketch on a sketch plane and angle of revolution. The sketch is revolved around the centerline.

(D). Revolved Cut Thin is a feature used to remove material by rotating a sketched profile around a centerline. the Right plane.

(E). Dome is a feature used to add a spherical or elliptical dome to a selected face.

2. A relation is a geometric constraint between sketch entities or between a sketch entity and a plane, axis, edge or vertex.

3. The Trim Entity sketch tool used to Trim unwanted geometry.

4. A Shell feature used to remove faces of a part by a specified wall thickness.

5. a center point, start point and end point.

6. The Hole Wizard feature is used to create specialized holes in a solid. The Hole Wizard types are: counter bore, counter sink, hole, straight Tap, Tapered Tap, Legacy Hole, counter bore Slot, counter sink Slot and Slot.

7. Spline is a complex sketch curve. Spline can have two or more points.

8. Circular Pattern is a feature that creates a pattern of features

or faces in a circular array about an axis. Use the circular pattern feature tool to create multiple instances of one or more feature that you can space uniformly around an axis.

9. View.

10. Right click Front Plane from Feature Manager. Click Show from the context tool bar. Hide unwanted planes in the Feature Manager if needed.

11. Convert Entities is a sketch tool that projects one or more curves onto the current sketch plane. Select an edge, loop, face, curve, or external sketch contour, set of edges, or set of sketch curves.

Offset Entities is a sketch tool utilized to create sketch curves offset by a specified distance. Select sketch entities, edges, loops, faces, curves, set of edge or set of curves.

12. The ends of mirrored lines, the center of arcs.

13. When you create mirrored entities, the Solid Work software applies a Symmetric relation between each corresponding pair of sketch points (the ends of mirrored lines, the centers of arcs). If you change a mirrored entity, its mirror image also change.

14. True.

15. False.

16. Feature Manager.

17. (A) Mirror Entities. (B) Spline. (C) Offset Entities. (D) Display/  
Delete Relations. (E) Parallelogramt. (F) Tex.

Answers.

- - A/ Number of line Types: 4.
  - b/ Types of lines : Dimension, extension, visible and
- 2.
  - a/ Number of Types: 3.
  - b/ Section, center and visible.
- 3. a/ Number of line Types: 7.
- b/ Visible, extension, dimension, hidden, break, center and centerline.

Page 1-37

- a/ Angle of Projection type: 4.
  - b/ Bottom view contains hidden line. Top view contains section lines.
  - c/ same, both contain hidden lines.
  - d/ the least amount of hidden lines is Height view.
- a/ Three views.
  - b/ they need only: Right, Top and Front views. Because these are the basic side of graph.
- a/ one view.
  - b/ Because only one shape.
- a/ Front view.
  - b/ Because the Front view show the most characteristic of the object.



- a/ Front view surfaces: B.  
B/ Top view surfaces: C.  
c/ Right view surfaces: D.

3.a/ Front view surfaces: G.

b/ Top view surfaces: B.

C/ Right view surfaces: E.

- a/ Front view surfaces: C.  
b/ Top view surfaces: A.  
c/ Right view surfaces: F.

7. a/ Front view surfaces: A.

b/ Top view surfaces: G.

c/ Right view surfaces: B.

8. a/ Front view surfaces: B.

b/ Top view surfaces: F.

c/ Right view surfaces: C.

9. a/ Front view surfaces: E.

b/ Top view surfaces: D.

c/ Right view surfaces: A.

10. a/ Top: E

b/ Front: A

c/ Right: I

11. a/ Top: J

b/ Front: E

c/ Right: G

page 1.53

1. Cartesian Coordinate System: specifies each point uniquely in a plane by a pair of numerical coordinates, which are the signed distances from the point to two fixed perpendicular directed lines, measured in the same unit of length. Each reference line is called a coordinate axis or just axis of the system, and the point where they meet is its origin.

2. Origin.

3. Most C A D systems use the Right-hand rule for coordinate system. To use the Right-hand rule - point your thumb of your right hand in the positive direction for the X axis and your index finger in the positive direction for the Y axis, your remaining fingers curl in the positive direction for the Z axis.

The Right-hand rule is also used to determine the direction of rotation.

4. Free hand sketching is important because visualizing and conceptualizing your idea that allows you to communicate that idea with others.

5. First Angle projection, the Top view is looking at the bottom of the part. is also used most of the world.

Third Angle projection, the Top view is looking at the Top of the part. is also used in America and Australia.

6. False.

7. When creating Orthographic views, it is common for one line type to overlap another line type. When this occurs, drawing conventions have established an order of precedence.

Example: a visible line type belongs in the same location as a hidden line type, since the visible features of a part (object lines) are represented by thick solid lines, they take precedence over all other lines.

8. False.

9. True.

10. The Glass Box method for Orthographic projection, a traditional method of placing an object in an imaginary glass box to view the six principle views.

11. True.

12. True.

13. True.

14. True. Example: in solid works, Break lines are displayed as short dashes or continuous solid lines, straight, curved or zig zag.

15. True. Example: for an automobile, the normal or operation position is on its wheels rather than on its roof or bumper.

Exercises 1.1 answer/ from top to bottom, and from left to right.

Top, Bottom

Back, Left, Front, Right

CHAPTER (2). Questions- page 2-38

1. Axonometric, Oblique and Perspective.

2. Isometric, Dimeric and Trimetric.

3. True.

4. Cavalier, Cabinet.

5. The two most characteristic feature of perspective are that objects are Drown: smaller as their distance from the observer increase and Foreshortened: the size of an object's dimensions along the line of sight are relatively shorter than dimensions across the line of sight.

6. True.

7. False.

8. True.

9. We already Drew.

10. We already Drew.

11. First angle projection the Top view is looking at the bottom of the part. And also used in Europe and most of the world. Third angle projection the Top view is looking at the Top of the part. And also used in America and Australia.

12. False.

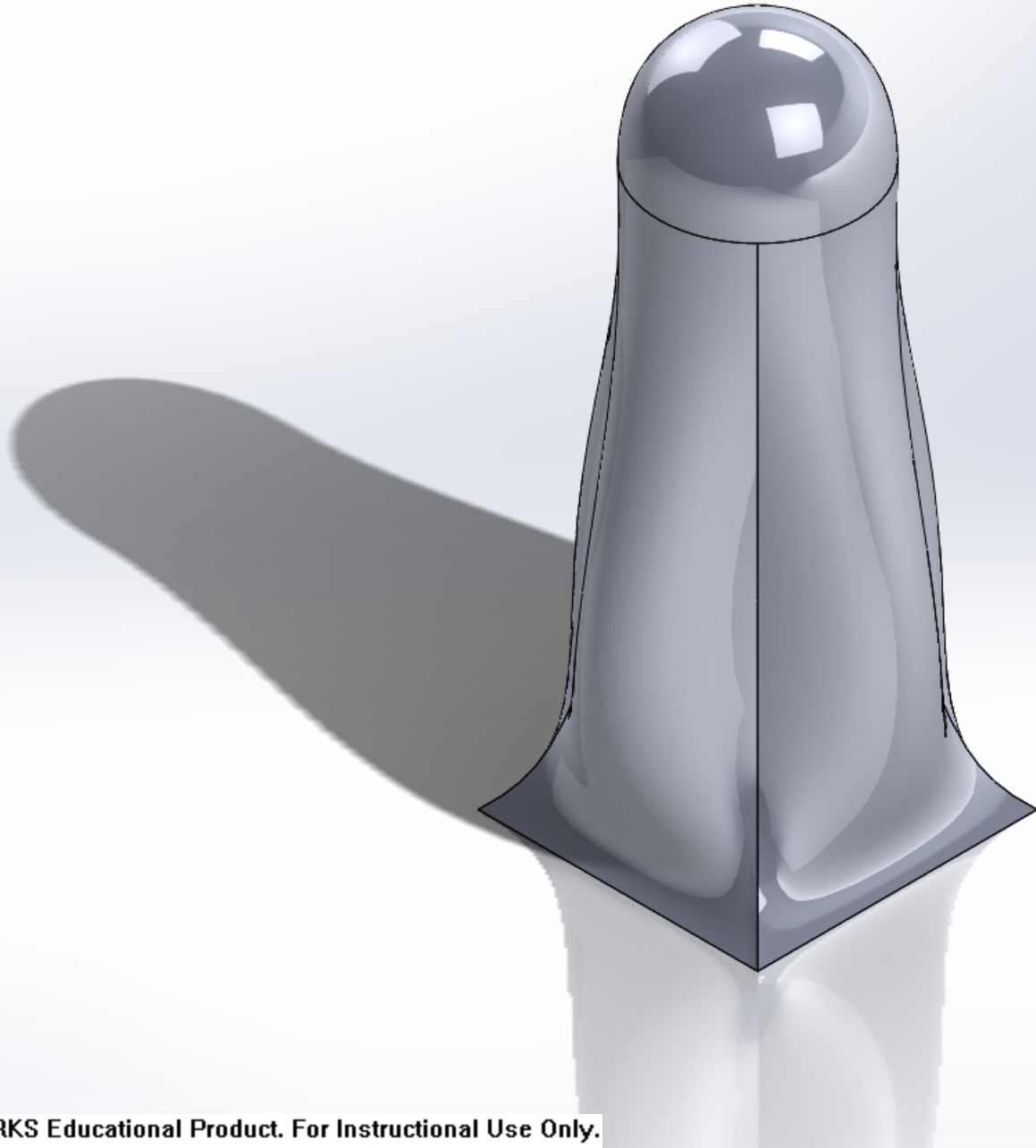
13. True.

14. True.

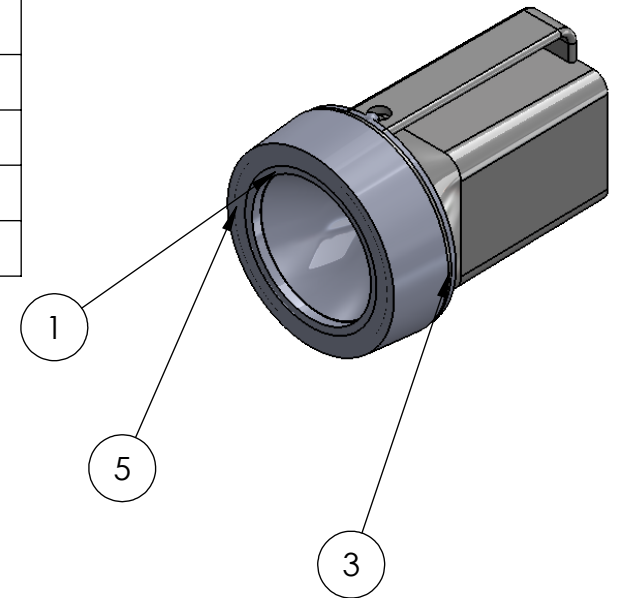
15. True. Break lines are displayed as short dashes or continuous solid lines, straight, curved or zig zag.

#### CHAPTER (3). Page3-44 Questions.

1. True.
2. (IPS) stand for inch, pond, second.
3. Square, round and elongated.
4. Rectangle.
5. Drilling, reaming or punching.
6. True.
7. "THRU" or "THRU ALL" in all upper case.
8. True.
9. Minutes ('). minutes, seconds.
10. "A"
11. "B"
12. Clearance, interference, transition.
13. "B" suffix.
14.  $\perp$ ,  $\sphericalangle$



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	HOUSING(1)	HOUSING FOR 6VOLT FLASHLIGHT	1
2	SWITCH		1
3	LENSCAP		1
4	O-RING		1
5	Lens		1
6	BULB	BULB FOR LENS	1
7	B99-B01		1
8	44-A26		1



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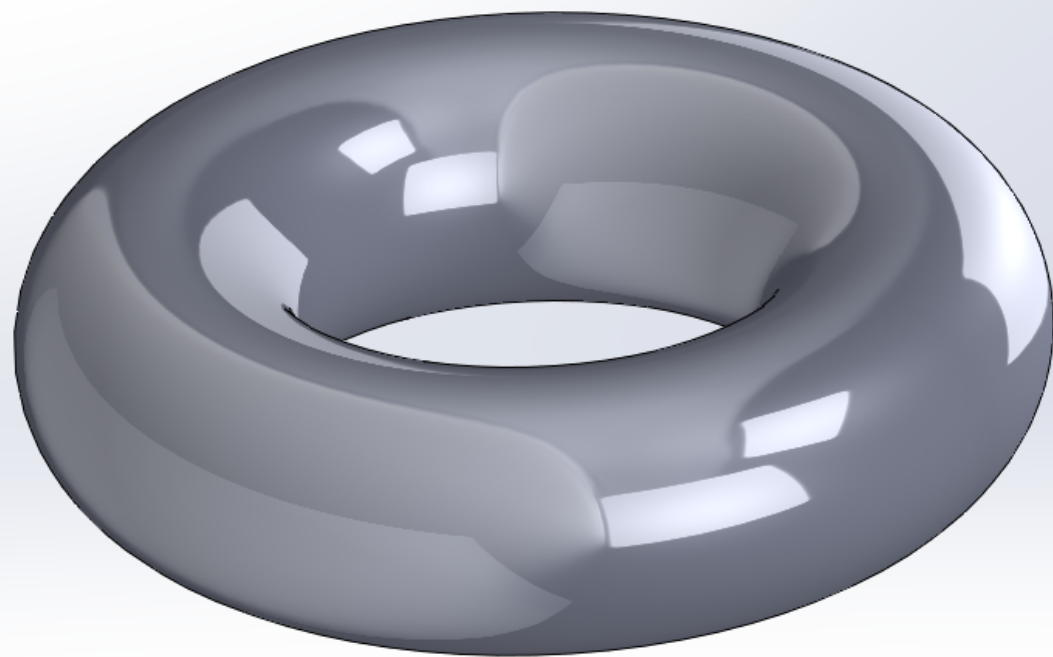
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# COLLEGEWIDE COURSE OUTLINE OF RECORD

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