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 soldies bodies within the same part document. A WRENCH consists of two cylindercal bodies. Each extrusion is a seprate body. The oval prrofile 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 byrevolutions. 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 useed to removes 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 toll 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 type 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 toll 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 toll bar. Hide unwanted planes in the Feature Manager if needed.

11. Convert Entities is a sketch toll 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 toll 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.