I. Basics & Introduction to SOLIDWORKS

- **Understanding the User Interface:**
  - FeatureManager Design Tree
  - Command Manager
  - View toolbar
  - Mouse buttons and keyboard shortcuts
  - File management
  - System requirements and settings
- **Parametric Design Concepts:**
  - Understanding design intent
  - File references
- **Basic Part Modeling Overview:**
  - Feature-based modeling

II. Sketching

- **2D Sketching:**
  - Sketch entities (lines, circles, rectangles, arcs, splines, polygons, ellipses, slots, points, text)
  - Sketch relations (geometric constraints like coincident, parallel, perpendicular, tangent, concentric, horizontal, vertical)
  - Dimensions (smart, horizontal, vertical, ordinate)
  - Sketch tools (trim, extend, offset, mirror, convert entities, linear/circular pattern, move, copy, rotate, scale, stretch)
  - Fully defining sketches
- **3D Sketching:**
  - Introduction to 3D sketching for specialized applications

III. Part Modeling

- **Basic Features:**
  - Extrude (Boss/Base and Cut)
  - Revolve (Boss/Base and Cut)
  - Fillet and Chamfer
  - Hole Wizard (standard holes, threads)
- **Reference Geometry:**
  - Creating planes, axes, coordinate systems, points
- **Intermediate/Advanced Features:**
  - Sweep (Boss/Base and Cut with guide curves)
  - Loft (Boss/Base and Cut with guide curves)
  - Rib
  - Draft
  - Shell
  - Patterns (linear, circular, sketch-driven, curve-driven, fill pattern, mirror)

- Configuration and Design Tables (for creating variations of a part)
- Material Library & Assigning Material
- Calculating mass and other geometric properties
- Boolean operations (Combine, Split)
- Editing and modifying features
- Multi-body solids

IV. Assembly Modeling

- **Getting Started with Assemblies:**
  - Assembly types (Top-Down, Bottom-Up)
  - Inserting components
  - FeatureManager Design Tree and symbols
- **Mates:**
  - Standard Mates (coincident, parallel, perpendicular, tangent, concentric, distance, angle, lock)
  - Advanced Mates (symmetric, width, path mate, linear/linear coupler, limit mate)
  - Mechanical Mates (cam, hinge, gear, rack pinion, screw, universal joint)
- **Manipulating Components:**
  - Moving and rotating components
  - Replacing components
  - Hiding components and controlling transparency
- **Assembly Tools:**
  - Interference detection and collision detection
  - Exploded views and explode lines
  - Bill of Materials (BOM)
  - Sub-assemblies
  - Smart Fasteners
  - Assembly patterns
  - Assembly visualization

V. Generating Detail Drawings

- **Drafting Overview:**
  - Drawing sheets and views
  - Adding drawing views (model view, projected views, standard 3 view, auxiliary views, section views, detail views, broken-out section, crop view, alternate position view)
- **Annotations and Dimensions:**
  - Smart dimensioning
  - Adding annotations, symbols, datum features, hole callouts, balloons
  - Sheet formats and templates
  - Tables (BOM, revision tables)

VI. Specialized Modules (often covered in advanced courses or as separate modules)

- **Surface Modeling:**
  - Introduction to surfacing tools (extrude, revolve, sweep, loft, offset surfaces)
  - Filleting, trimming, extending, knitting surfaces
  - Creating and manipulating complex organic shapes
- **Sheet Metal Design:**
  - Sheet metal features (base flange, edge flange, miter flange, hem, jog, sketched bend)
  - Forming tools
  - Converting parts to sheet metal
  - Flattening and unfolding for manufacturing
- **Weldments:**
  - 3D sketching for weldment structures
  - Structural members
  - Trimming and extending members
  - Gussets, end caps
  - Cut lists and weld beads
- **Mold Design:**
  - Core and Cavity design
  - Draft analysis, shut-off surfaces, parting lines
- **Routing (Electrical/Piping and Tubing):**
  - Creating routes for electrical cables, wires, pipes, and tubes
- **Simulation (FEA/CFD):**
  - SOLIDWORKS Simulation (basic structural analysis, stress, displacement)
  - SOLIDWORKS Flow Simulation (fluid dynamics, thermal analysis)
  - SOLIDWORKS Plastics (mold flow analysis)
- **SOLIDWORKS PDM (Product Data Management):**
  - Managing and controlling design data
- **SOLIDWORKS CAM:**
  - Introduction to computer-aided manufacturing
- **SOLIDWORKS Visualize/PhotoView 360:**
  - Creating photorealistic renderings

This syllabus provides a solid foundation for anyone looking to learn and master SOLIDWORKS, from beginners to those aiming for professional certification.