SUBJECT : MTS 220 Solid Modelling

CREDIT HOURS : 0-1

CONTACT HOURS : 3 Hours per Week

TEXT BOOKS :
1. PTC Design & Technology in Schools Curriculum
2. Getting Started with Pro/ENGINEER Wildfire

REFERENCE BOOKS : Any Pro-Engineer Bool/Tutorials

MODE OF TEACHING : Lectures/Hands on Exercises on computers

COURSE OBJECTIVES: The main objectives are to provide students with a conceptual understanding of the principles of CAD systems, the implementation of these principles, and its connections to CAM and CAE systems. The generic aspect of CAD software systems will be discussed. A large portion of the students' time will be spent independently in the computer labs learning the details of design and analysis related to the product realization process. The software system used in teaching/learning is Pro/ENGINEER Wildfire.

COURSE OUTCOMES:

At the completion of the course students will be able to

1. Review the process of engineering problem solving
2. Review computer and graphics technologies
3. Introduce methodology in design
4. Learn the geometric description in CAD/CAM
5. Learn the geometric modeling
6. Understand the basics of FEA
7. Learn about applications of CAD/CAM technology

TOPICS COVERED:

1. Getting Acquainted with the Pro/E Interface, Pro/Engineer Wildfire 4 User Interface
2. Sketcher
3. Datum features.
4. Extruded Protrusions & cuts
5. Holes, Rounds and Chamfers
6. Shells, Ribs
7. Feature Modification and Manipulation, Patterns and Copies
8. Revolved Protrusions & cuts
9. Sweeps and Swept Protrusions & cuts
10. Blended Protrusion & Swept blends
11. Assembly Modeling
12. Sheet Metal Design
13. Surface Modeling, Style features.
14. Motion, Structure and Thermal Analyses

List of Experiments

Week	Details
1	**Lab 1: Getting Acquainted with the Pro/E Interface** • Understand CAD terminology • Understand Pro/Engineer Wildfire concepts and conventions • Be able to navigate the Pro/Engineer Wildfire 4.0 Interface • Understand how to interact with Pro/ENGINEER • Keyboard and Mouse Control Keys in Pro/Engineer • Basic features of Pro/E including how to access part files, manipulate the display, and orient parts.
2	**Lab 2: Creating basic shapes and transforming them to 3D objects utilizing Extrusion** • Extrusions are one of the most basic ways to design a part in Pro/E, and sketching is an important part of the design process • Create a simple part using an extruded protrusion. Modify the part using a cutting operation.

3	**Lab 3: Holes, Rounds and Chamfers** • A variety of geometric shapes and constructions can be designed automatically with Pro/E, including holes, rounds, and chamfers. • The Hole option creates many types of holes, including straight holes, sketched holes, and holes for standard fasteners. T • The Round option creates a fillet or a round on an edge that is a smooth transition with a circular profile between two adjacent surfaces. • The Chamfer option creates a beveled surface at the intersection of edges. • How holes are placed by locating their centers with two linear dimensions, along a know axis, at a point, or on a radial direction
4	**Lab 4: Shells, Ribs and Datum Planes** • Datum planes are used as sketching surfaces and references for creating and construction features. • Learn to use datum planes to create a rib. • Learn how to create several basic features.
5	**Lab 5: Feature Modification and Manipulation. Revolves, Patterns and Copies** • Modify and manipulate features in Pro/E. • Learn how to use the model tree to modify various aspects of a feature • The Revolve option is useful for designing circular parts and features. • Patterns and copies allow multiple instances of a feature to be created with little effort • Create a pattern of holes and a copy of this pattern.
6	**Lab 6: Sweeps and Blends** • Sweeping and blending are useful techniques for designing parts which may be difficult to model with extrusions or revolves. • Create a constant cross-section U-shaped part using a sweep and a vase using a blend

	• Create a sweep protrusion along a sketch trajectory • Create a Helical sweep • Create an intersect and sweep (2D sweep)
7	**Lab 7: Assembly Modeling** • Pro/E's assembly module allows parts to be grouped into assemblies or subassemblies to model a complete part or mechanism. • Learn how to create assemblies, apply constraints between parts, change view properties of parts, and create exploded views. • Make a pulley mechanism assembly.
8	**Lab 8: Cam operated toy project** • Be aware of: 1. Physical and virtual 3D modeling techniques 2. Simple cam mechanisms and toys that use these • Understand : 1. The concepts behind simple cam mechanisms 2. The principles of 3D solid modeling software • Be able to: 1. Use simple workshop techniques to assemble parts into a working mechanism 2. Use Pro/ENGINEER to assemble parts into a working mechanism 3. Use Pro/Engineer to create a new component using sketching and extrude
9	**Lab 9: Sheet Metal Design and Surface Modelling** • Sheet metal is a commonly used material for the design of engineering systems • Learn to design sheet metal parts containing multiple walls, bends, cuts, and holes. Learn how to create a flat pattern of the part • Surface models are created by defining boundary curves, adding surfaces between curves, and combining surfaces to create parts

	• Learn how to create surfaces, create models from surfaces, and add features to surface models while designing a telephone receiver.
10	**Lab 10: Pro/Engineer Mechanica Analysis** • Learn how to do static, kinematic, kinetic analyses of assembly mechanism. • Learn how to do FEA (Finite Element Analysis) stress analysis • Learn how to do thermal analysis • Design Inverted Pendulum and undertake motion, structure and thermal analyses
11	**Lab 11: Robotic Arm Project**