CHAPTER 1 – INTRODUCTION

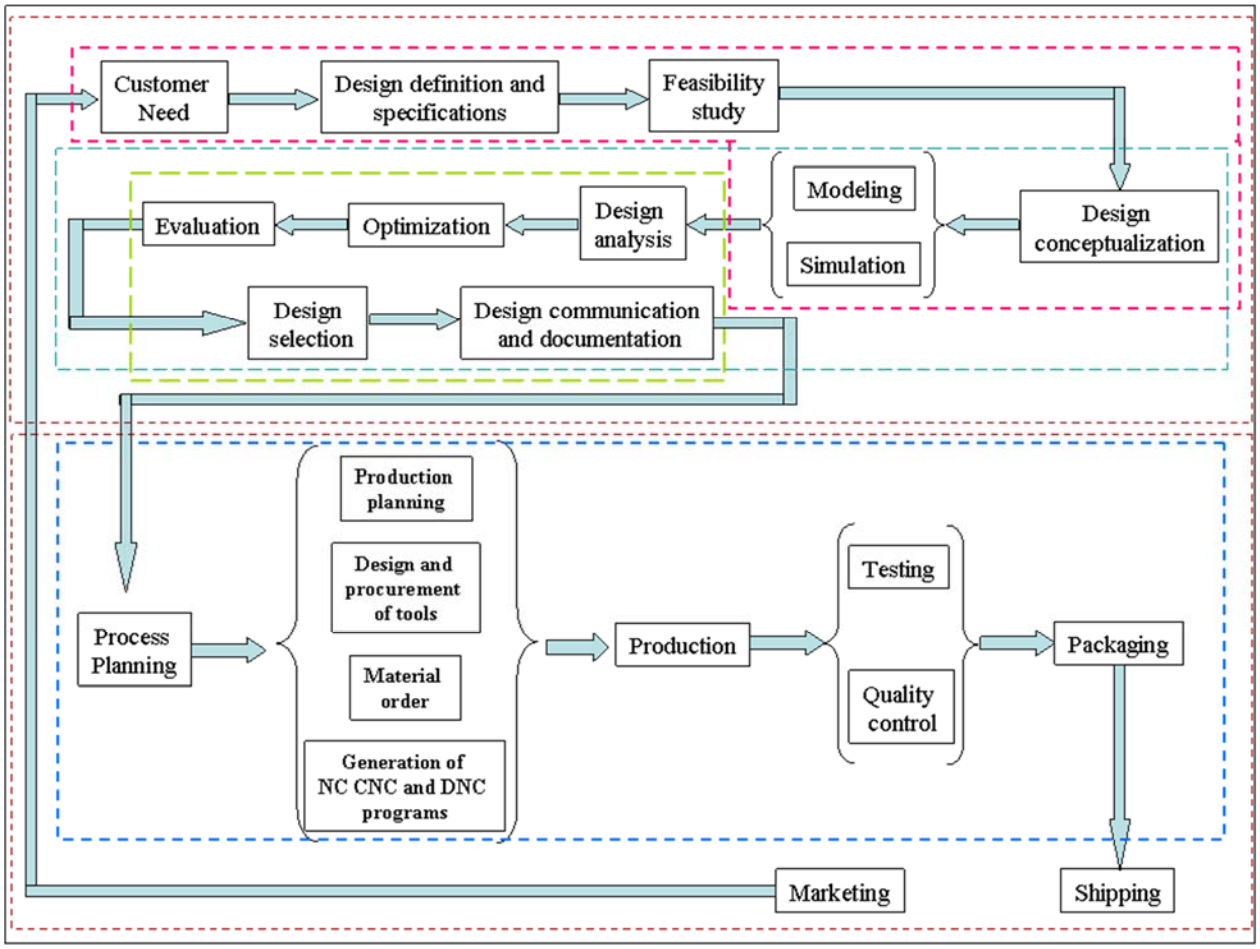
The modern manufacturing environment can be characterized by the paradigm of delivering products of increasing variety, smaller batches and higher quality in the context of increasing global competition. Industries cannot survive worldwide competition unless they introduce new products with better quality, at lower costs and with shorter lead-time. There is intense international competition and decreased availability of skilled labor. With dramatic changes in computing power and wider availability of software tools for design and production, engineers are now using *Computer Aided Design* (CAD), *Computer Aided Manufacturing* (CAM) and *Computer Aided Engineering* (CAE) systems to automate their design and production processes. These technologies are now used every day for all sorts of different engineering tasks. Below is a brief description of how CAD, CAM, and CAE technologies are being used during the product realization process.

1.1 PRODUCT REALIZATION PROCESS

The product realization process can be roughly divided into two phases: design and manufacturing. The design process starts with identification of new customer needs and design variables to be improved, which are identified by the marketing personnel after getting feedback from the customers. Once the relevant design information is gathered, design specifications are formulated. A feasibility study is conducted with relevant design information and detailed design and analyses are performed. The detailed design includes design conceptualization, prospective product drawings, sketches and geometric modeling. Analysis includes stress analysis, interference checking, kinematics analysis, mass property calculations and tolerance analysis, and design optimization. The quality of the results obtained from these activities is directly related to the quality of the analysis and the tools used for conducting the analysis.

The manufacturing process starts with the shop-floor activities beginning from production planning, which uses the design process drawings and ends with the actual product. Process planning includes activities like production planning, material procurement, and machine selection. There are varied tasks like procurement of new tools, NC programming and quality checks at various stages during the production process. Process planning includes planning for all the processes used in manufacturing of the product. Parts that pass the quality control inspections are assembled functionally tested, packaged, labeled, and shipped to customers.

A diagram representing the Product Realization Process (*Mastering CAD/CAM*, by Ibrahim Zeid, McGraw Hill, 2005) is shown below.



1.2 BRIEF HISTORY OF CAD/CAM DEVELOPMENT

The roots of current CAD/CAM technologies go back to the beginning of civilization when engineers in ancient Egypt recognized graphics communication. Orthographic projection practiced today was invented around the 1800s. The real development of CAD/CAM systems started in the 1950s. CAD/CAM went through four major phases of development in the last century. The 1950s was known as the era of interactive computer graphics. MIT's Servo Mechanisms Laboratory demonstrated the concept of numerical control (NC) on a three-axis milling machine. Development in this era was slowed down by the shortcomings of computers at the time. During the late 1950s the development of Automatically Programmed Tools (APT) began and General Motors explored the potential of interactive graphics.

The 1960s was the most critical research period for interactive computer graphics. Ivan Sutherland developed a sketchpad system, which demonstrated the possibility of creating drawings and altercations of objects interactively on a cathode ray tube (CRT). The term CAD started to appear with the word ‘design’ extending beyond basic drafting concepts. General Motors announced their DAC-1 system and Bell Technologies introduced the GRAPHIC 1 remote display system.

During the 1970s, the research efforts of the previous decade in computer graphics had begun to be fruitful, and potential of interactive computer graphics in improving productivity was realized by industry, government and academia. The 1970s is characterized as the golden era for computer drafting and the beginning of ad hoc instrumental design applications. National Computer Graphics Association (NCGA) was formed and Initial Graphics Exchange Specification (IGES) was initiated.

In the 1980s, new theories and algorithms evolved and integration of various elements of design and manufacturing was developed. The major research and development focus was to expand CAD/CAM systems beyond three-dimensional geometric designs and provide more engineering applications.

The present day CAD/CAM development focuses on efficient and fast integration and automation of various elements of design and manufacturing along with the development of new algorithms. There are many commercial CAD/CAM packages available for direct usages that are user-friendly and very proficient.

Below are some of the commercial packages in the present market.

- Solid Edge, AutoCAD, Inventor and TurboCAD are some affordable CAD software systems.
- NX, Pro-E, CATIA and SolidWorks are high-end modeling and designing software systems that are costlier but more powerful. These software systems also have computer aided manufacturing and engineering analysis capabilities.
- Onshape and Fusion 360 are cloud based CAD software, which provide CAD capabilities via user’s browser.

- ANSYS, ABAQUS, NASTRAN and COMSOL are packages mainly used for CAE purposes.

1.3 DEFINITION OF CAD/CAM/CAE

Following are the definitions of some of the terms used in this tutorial.

1.3.1 Computer Aided Design – CAD

CAD is technology concerned with using computer systems to assist in the creation, modification, analysis, and optimization of a design. Any computer program that embodies computer graphics and an application program facilitating engineering functions in design process can be classified as CAD software.

The most basic role of CAD is to define the geometry of design – a mechanical part, a product assembly, an architectural structure, an electronic circuit, a building layout, etc. The greatest benefits of CAD systems are that they can save considerable time and reduce errors caused by otherwise having to redefine the geometry of the design from scratch every time it is needed.

1.3.2 Computer Aided Manufacturing – CAM

CAM technology involves computer systems that plan, manage, and control the manufacturing operations through computer interface with the plant's production resources.

One of the most important areas of CAM is numerical control (NC). This is the technique of using programmed instructions to control a machine tool, which cuts, mills, grinds, punches or turns raw stock into a finished part. Another significant CAM function is in the programming of robots. Process planning is also a target of computer automation.

1.3.3 Computer Aided Engineering – CAE

CAE technology uses a computer system to analyze the functions of a CAD-created product, allowing designers to simulate and study how the product will behave so that the design can be refined and optimized.

CAE tools are available for a number of different types of analyses. For example, kinematic analysis programs can be used to determine motion paths and linkage velocities in mechanisms. Dynamic analysis programs can be used to determine loads and displacements in complex assemblies such as automobiles. One of the most popular methods of analyses is using a Finite

Element Method (FEM). This approach can be used to determine stress, deformation, heat transfer, magnetic field distribution, fluid flow, and other continuous field problems that are often too tough to solve with any other approach.

1.4. SCOPE OF THIS TUTORIAL

This tutorial is written for students and engineers who are interested in learning how to use NX 12 for designing mechanical components and assemblies. Learning to use NX 12 will also be valuable for learning how to use other CAD systems such as PRO-E and CATIA. This tutorial provides a systematic approach for learning NX 12.

Chapter 2 includes the NX 12 essentials from starting a session to getting familiar with the NX 12 layout by practicing basic functions such as Print, Save, and Exit. It also gives a brief description of the Coordinate System, Layers, various toolboxes and other important commands, which will be used in later chapters.

Chapter 3 presents the concept of sketching. It describes how to create sketches and to give geometric and dimensional constraints. This chapter is very important since present-day components are very complex in geometry and difficult to model with only basic features.

The actual designing and modeling of parts begins with **chapter 4**. It describes different features such as reference features, swept features and primitive features and how these features are used to create designs. Various kinds of feature operations are performed on features.

You will learn how to create a drawing from a part model in **chapter 5**. In this chapter, we demonstrate how to create a drawing by adding views, dimensioning the part drawings, and modifying various attributes in the drawing such as text size, arrow size and tolerance.

Chapter 6 teaches the concepts of Assembly Modeling and its terminologies. It describes Top-Down modeling and Bottom-Up modeling. We will use Bottom-Up modeling to assemble components into a product.

Chapter 7 introduces free-form modeling. The method of modeling curves and smooth surfaces will be demonstrated.

Chapter 8 is capsulated into a brief introduction to Design Simulations available in NX 12 for the Finite Element Analysis.

Chapter 9 will be a real-time experience of implementing a designed model into a manufacturing environment for machining. This chapter deals with generation, verification and simulation of Tool Path to create CNC (Computer Numerical Codes) to produce the designed parts from multiple axes and even advanced CNC machines.

The examples and exercise problems used in each chapter are so designed that they will be finally assembled in the chapter. Due to this distinctive feature, you should save all the models that you have generated in each chapter.