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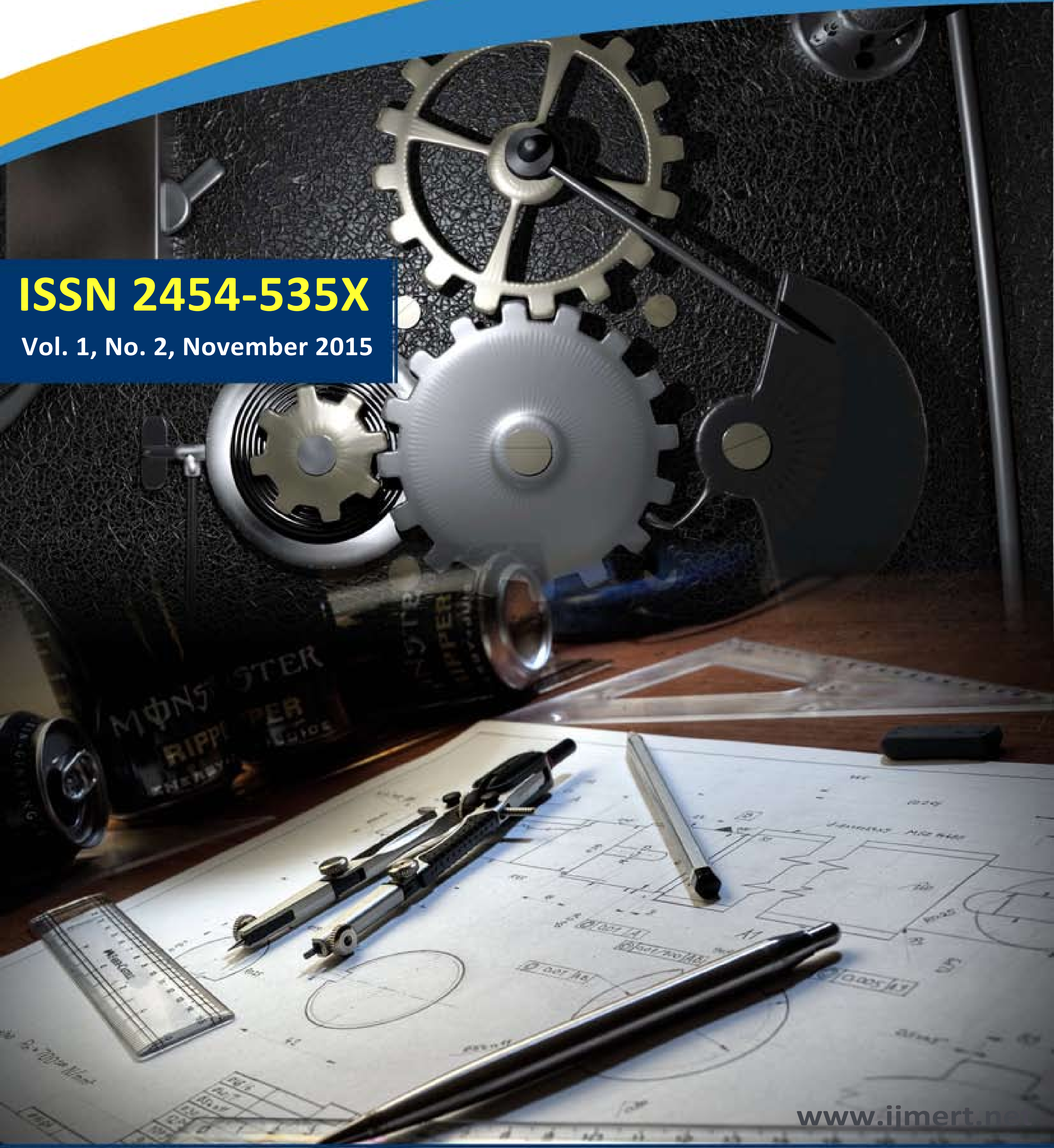

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Research Paper

DESIGN AND MANUFACTURING OF AIR FRAME MISSILE

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The airframe missile is a small, lightweight, infrared homing surface-to-air missile in use by the American, German, south Korean, Greek, Turkish, Saudi and Egyptian navies. It was intended originally and used primarily as a point-defense weapon against anti-ship cruise missiles. The missile is so-named because it rolls around its longitudinal axis to stabilize its flight path, much like a bullet fired from a rifled barrel. Computer-aided manufacturing CAM reduces waste and energy for enhanced manufacturing and production efficiency via increased production speeds, raw material consistency and more precise tooling accuracy. The aim of the paper is to create a 3D model in the CAD software. Develop different manufacturing process plans by changing the work holding systems, tool paths, cutting tools, etc., optimizing the process plan for high surface finish and less machining time. In this paper we made a design by using NX7.5 CAD software and manufacturing missile shield.

Keywords: Airframe missile, 3D model, CAD/CAM, NX7.5

INTRODUCTION

The paper deals with the design and tool path generation for "airframe missile" component using CAM software ('UGNX-7.5' which is a CAD/CAM software used to generate part program by designing and feeding the geometry of the component) and defining the proper tool path and thus transferring the generated part program to the required CNC machine with the help of DNC lines. Then the program is executed with suitable requirements.

There are many factors responsible for rejection and reworks, such as human errors, machine errors, process planning, material errors, etc. Due to this thin wall thickness there is more chance of rejections and reworks. Iterations/experiments cannot be made on a CNC machine, because of its high operating cost.

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The latest CAM software introduced includes the new NX 7.5 software, which has important features like 2D, 3D and surface modeling. The component can be either designed on this software or can be retrieved from any other CAD software. Then sequence of programs such as modeling the component, selection of tools according to the sequence of operations and sizes, generating the tool path, at last the generated NC part program is verified and sent to the required CNC machine to manufacture the particular component. Finally the required surface finish has been obtained by machining the component at optimum speeds and feeds and the cost of machining is also optimized by choosing optimal machining process and machine tools.

UNIGRAPHICS

NX is one of the world's most advanced and tightly integrated CAD/CAM/CAE product development solutions. Spanning the entire range of product development, NX delivers immense value to enterprises of all sizes. It simplifies complex product designs, thus speeding up the process of introducing products to the market.

The NX software integrates knowledge-based principles, industrial design, geometric modeling, advanced analysis, graphic simulation, and concurrent engineering. The software has powerful hybrid modeling capabilities by integrating constraint-based feature modeling and explicit geometric modeling. In addition to modeling standard geometry parts, it allows the user to design complex free-form shapes such as airfoils and manifolds. It also merges solid and surface modeling techniques into one powerful tool set. Our previous efforts to prepare the NX self-guiding tutorial were funded by the National Science Foundation's Advanced Technological.

Figure 1

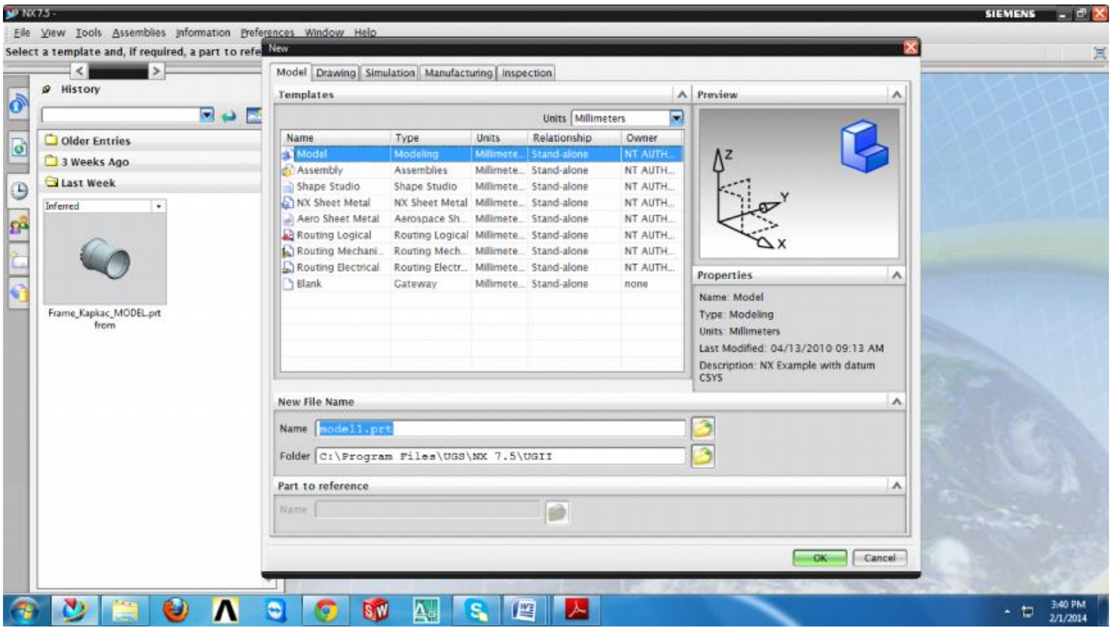
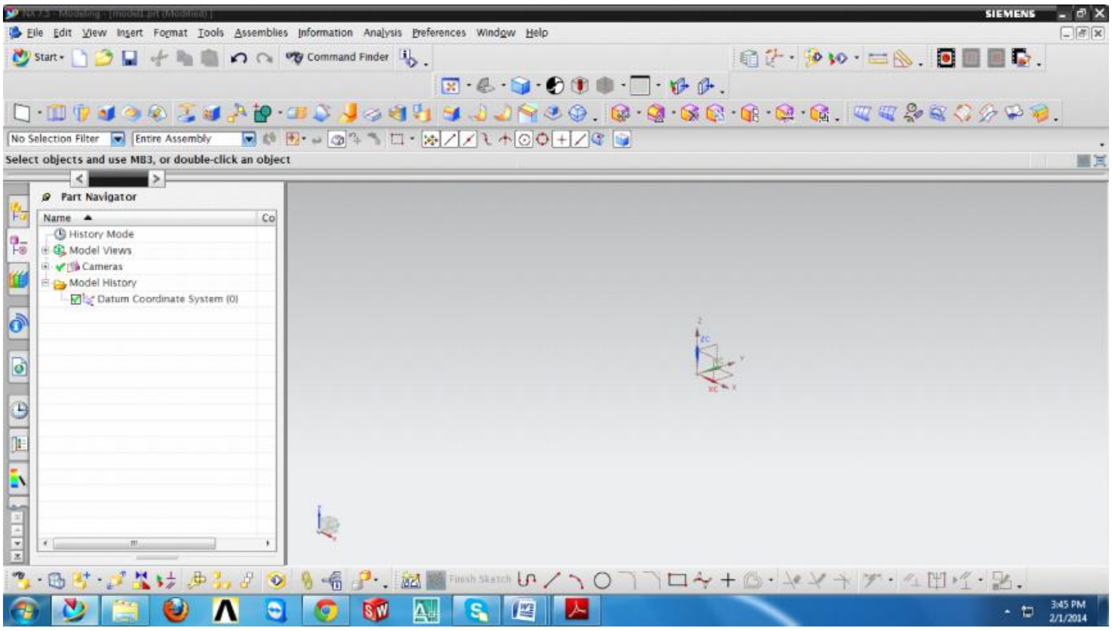


Figure 2



Once the new file has been created, the NX modelling interface will open. Like most modern PLM tools, the interface for NX contains numerous icons, lists, text prompts and other features that can be incredibly overwhelming. For now, we will focus on the sketching tools, part navigator, viewer and menu.

DRAWING IN NX

NX, like most modern PLM tools, is feature-based. That means you build up a component from a set of features that are added in sequence. This sequence, and the details of each feature, can be edited later if the design needs to be changed. Features can be removed, inserted or modified to create almost any solid part imaginable. This allows a precise and highly editable method for describing solid components and differs from the 3D freeform modeling approach (freeform modeling is used in computer graphics applications such as making models for video games). The features you add to a part will appear in the Part Navigator under Model History. For a new part, the only thing appearing in the model history should be a Datum Coordinate System. This is the 3D Cartesian coordinate system used in the part and includes an origin location and coordinate axes. These are shown with the coordinate system indicator at the center of the Viewer. The general workflow for creating a part in NX is to create a sketch, extrude or revolve the sketch and then add features to the resulting solid part. Sketches are 2D (mostly) drawings that are used to define a cross section of the part. This cross section can be extruded (pushed into or pulled out of the plane of the sketch) and rotated (wrapped around a axis) to form a 3D solid part. To create a Sketch. Click the Sketch button in the sketching toolbar. This will open a new dialog for defining the sketch. The first step in defining the Sketch is to set the sketching plane. This is the 2D plane that the sketch will be drawn onto. This plane can be derived from the existing Datum Coordinate System or can be based on geometry from an existing part (you can sketch on the surface of a part). For now, start the sketch on the XY plane. To select the XY plane, move the cursor to the skewed lines connecting the x and y axis lines on the coordinate system indicator.

Increasing complexity of products, development processes and design teams is challenging companies to find new tools and methods to deliver greater innovation and higher quality at lower cost. Leading-edge technology from Siemens PLM software delivers greater power for today's design challenge. From innovative Synchronous Technology that unites parametric and history-free modeling, to NX Active Mockup for multi-CAD assembly design, NX delivers breakthrough technology that sets new standards for speed, performance, and ease of use.

NX automates and simplifies design by leveraging the product and process knowledge that companies gain from experience and from industry best practices. It includes tools that designers can use to capture knowledge to automated repetitive tasks. The result is reduced cost and cycle time and improved quality.

CANISTER THEORY

A lamina is a thin layer of a composite material that is generally of a thickness on the order of 0.005 in. (0.125 mm). A laminate is constructed by stacking a number of such laminae in the direction of the lamina thickness. Mechanical structures made of these laminates, such as a leaf spring suspension system in an automobile, are subjected to various loads, such as bending and twisting. The design and analysis of such laminated structures demands knowledge of the stresses and strains in the laminate. Also, design tools, such as failure theories, stiffness models, and optimization algorithms, need the values of these laminate stresses and strains. Generally, a laminate does not consist only of unidirectional laminae because of their low stiffness and strength properties in the transverse direction. Therefore, in most laminates, some laminae are placed at an angle. The laminate theory is used for calculation of engineering constants of laminae placed at different orientations.

The present canister is designed based on the properties of epoxy resin and glass fiber.

Table 1: Mechanical Properties of Glass Fiber (Reinforcement Phase)

Specific gravity	2.5
Young's modulus (E_f)	85 GPa
Ultimate tensile strength	1550 MPa
Poisson's ratio (v_f)	0.2

Note: Where E_f = Young's Modulus of glass fiber at 0 orientation
v_f = Poisson's Ratio

Table 2: Mechanical Properties of Epoxy (Matrix Phase)

Specific gravity	1.28
Young's modulus (E_m)	3.792 GPa
Ultimate tensile strength	82.74 MPa

Note: Where E_m = Young's Modulus of Epoxy

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The properties are tabulated in Tables 1 and 2. The properties of glass/epoxy lamina placed at different fiber orientation are calculated using laminate theory as shown below:

Then

Longitudinal young's modulus $(E_1) = (E_f)(V_f) + (E_m)(V_m)$

Transverse young's modulus $(1/E_2) = (V_f/E_f) + (V_m/E_m)$

Major poisson's ratio $(v_{12}) = (v_f)(V_f) + (v_m)(V_m)$

Minor poisson's ratio $(v_{21}) = v_{12}*(E_2/E_1)$

Shear modulus of fibre $(G_f) = E_f/2(1 + v_f) =$ 35.42 Gpa

Shear modulus of matrix $(G_m) = E_m/2(1 + v_m) =$ 1.308 Gpa

Shear modulus $(1/G_{12}) = (V_f/G_f) + (V_m/G_m)$

By using above relations we get the following values

MISSILE PROPERTIES

The launching of a missile involves creating a internal pressure of 30 bar by using chemical reactions inside the canister due to which missile is ejected. Here we are considering the specifications of canister AGNI V for the design. It is designed for resisting a pressure of 30 bar internal pressure. Since we are considering that missile is in submarine, an external pressure is also created. As we know for every 2 meters depth a pressure of 1 bar is developed, presently we are considering the canister is at 10 meters depth and a pressure of 5 bar is acting externally. So our design must

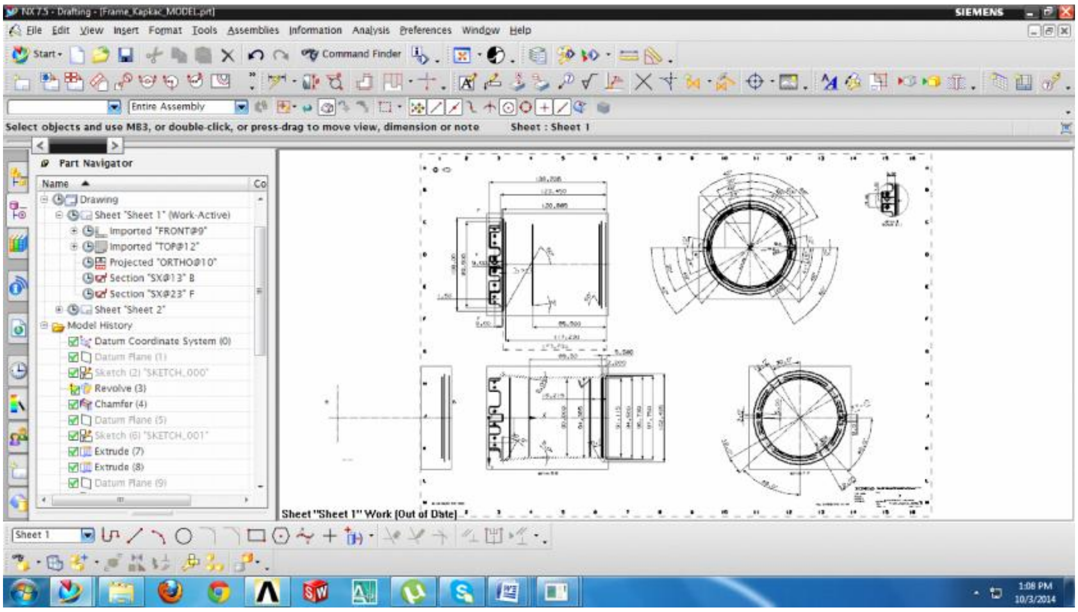
Table 3: Engineering Constants for Various Proportions of Glass/Epoxy

Composition	Longitudinal (E_1)	Transverse (E_2)	Major Poisson Ratio (v_{12})	Minor Poisson Ratio (v_{21})	Shear Modulus (G_{12})
V_f=0.35, V_m=0.65	32.214	5.71	0.265	0.04	1.97
V_f=0.40, V_m=0.60	36.275	6.72	0.26	0.048	2.127
V_f=0.45, V_m=0.55	40.335	6.666	0.255	0.042	2.309
V_f=0.50, V_m=0.50	44.396	7.299	0.25	0.041	2.525
V_f=0.55, V_m=0.45	48.456	8	0.245	0.04	2.78
V_f=0.60, V_m=0.40	52.516	8.928	0.24	0.04	3.105
V_f=0.65 V_m=0.35	56.577	10.101	0.235	0.041	3.508

Table 4: Typical Properties of Unidirectional Lamina

Property	Symbol	Units	Glass/Epoxy
Fibre volume fraction	V_f		0.45
Longitudinal elastic modulus	E_1	GPa	38.6
Transverse elastic modulus	E_2	GPa	8.27
Major Poisson's Ratio	V_{12}		0.26
Shear Modulus	G_{12}	GPa	4.14
Ultimate longitudinal tensile strength	ult	MPa	1062
Ultimate longitudinal compressive strength	ult	MPa	610
Ultimate Transverse tensile strength	)ult	MPa	31
Ultimate Transverse Compressive strength	)ult	MPa	118
Ultimate in-planar shear strength	$(\tau_{12})_{ult}$	MPa	72

Figure 3



resist an external pressure of 5 bar and internal pressure of 30 bar.

INPUT FOR THE PROJECT

2D Drawing

A 2D drawing is used to design a 3D model for our component using Unigraphics NX 7.5 CAD software.

Below shows the 2D drawings of the Airframe Missile Component with all the required dimensions and GD and T representations the suits the best for manufacturing the component without any errors.

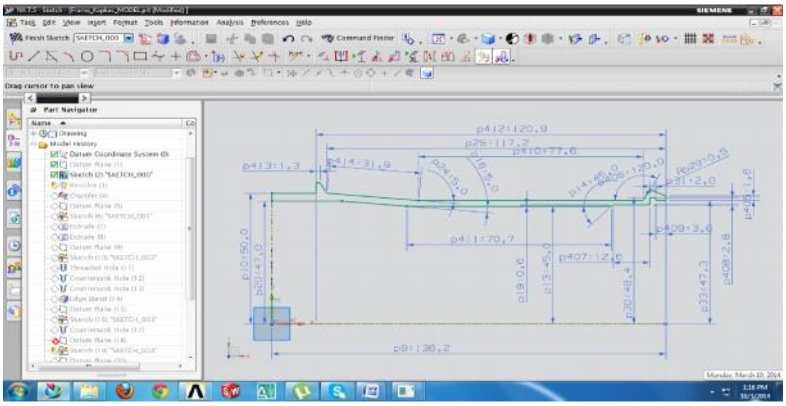
STEPS INVOLVED IN 3D MODELING

Sketching

Below is the sketch required to obtain the 3D model of the Airframe Missile Component from the above 2D drawing 3D model is designed by using cad NX 7.5 software.

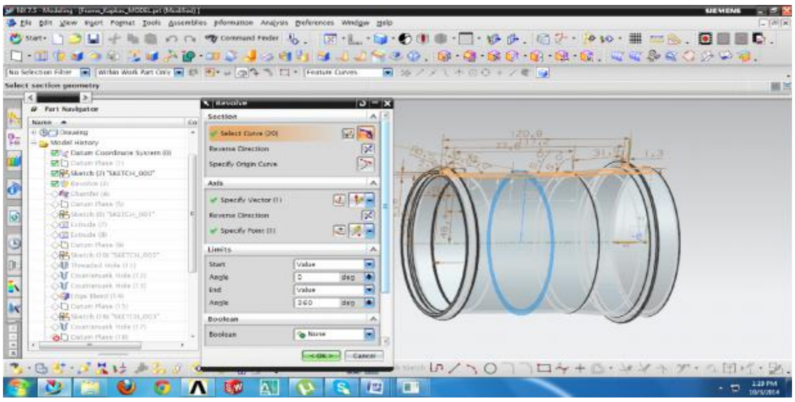
Below image shows the sketch of the Airframe Missile Component.

Figure 4



Note: Procedure to draw the above sketch
Insert → sketch in task environment → select plane → ok.
Insert → curve → profile.

Figure 5: Revolve



Note: Procedure to revolve the above sketch.
Insert → design features → revolve.
Select curve → specify vector → Boolean operation (none) → ok.

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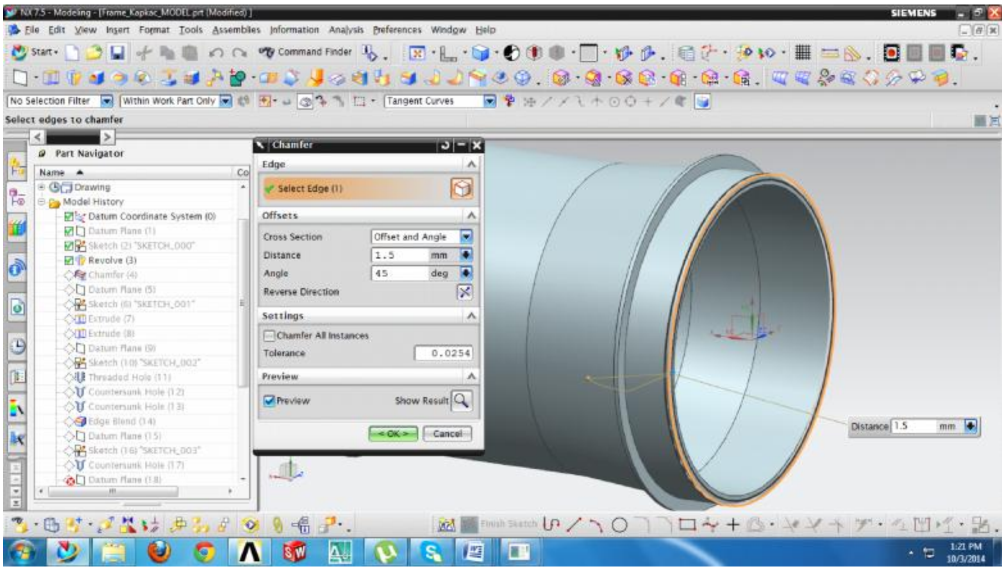
By using profile curve we will get the 2D design of Airframe Missile Component.

Below image shows the revolve option.

By using revolve command we convert sketch from 2D to 3D only for axis symmetry bodies.

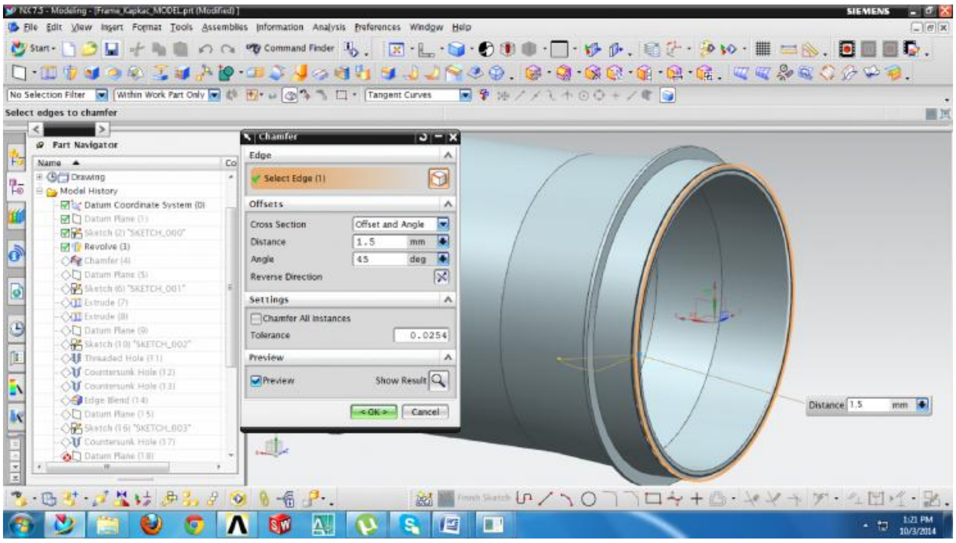
Below image shows the Chamfer option.

Figure 6: Chamfer



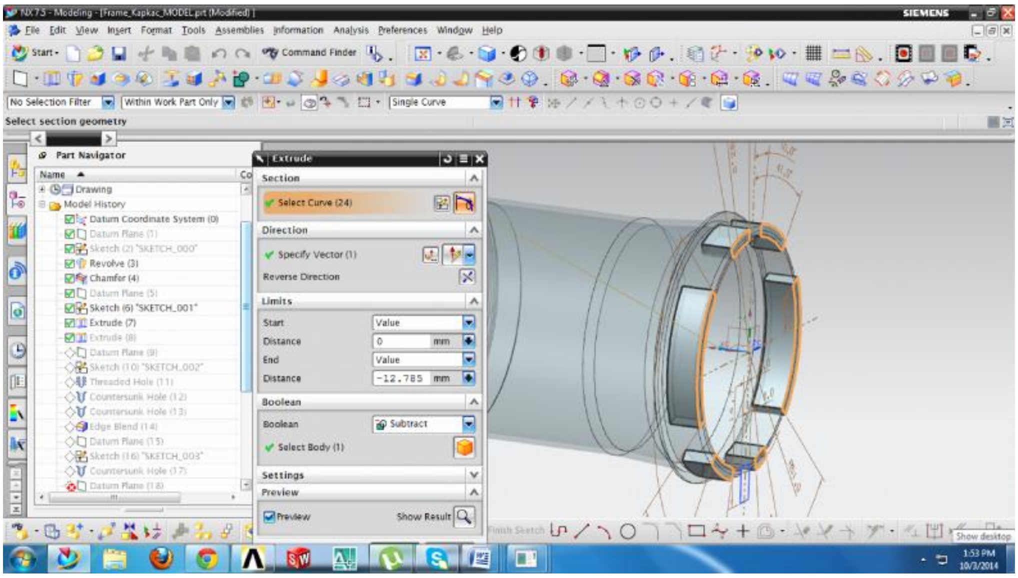
Note: Creates round edges between faces.
Insert → detail features → chamfer.
Select edge → specify distance → ok

Figure 7



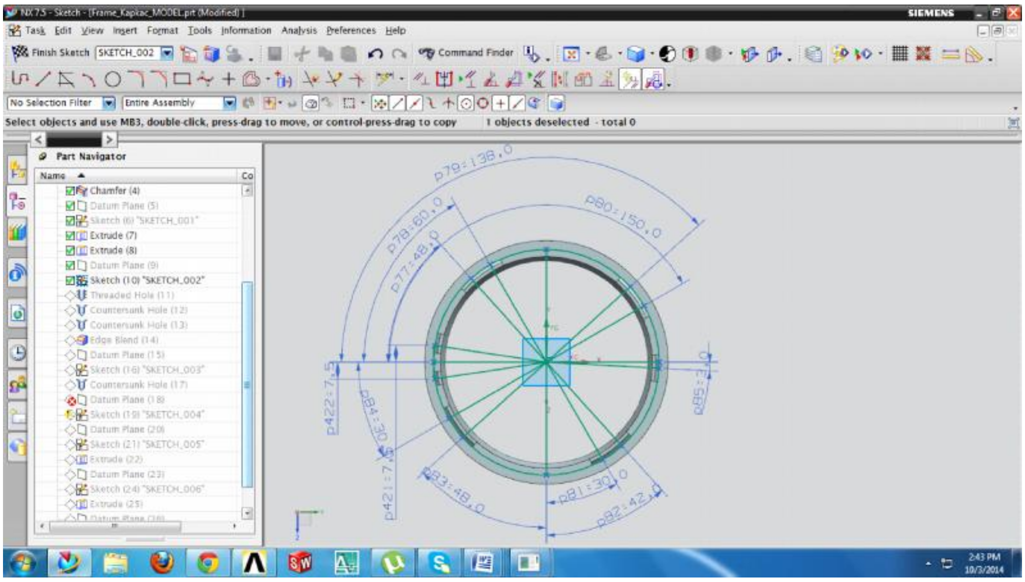
Note: Procedure to draw the above sketch
Insert → sketch in task environment → select plane → ok.
Insert → curve → profile.

Figure 8: Extrude



Note: Extrude command is used to create a body by sweeping a 2D or 3D section of curves, Edges, sketches in a specified Direction.
Insert → design features → extrude.
Select curve → specify vector → Boolean operation (subtract) → ok

Figure 9



Note: Procedure to draw the above sketch
Insert → sketch in task environment → select plane → ok.
Insert → curve → profile

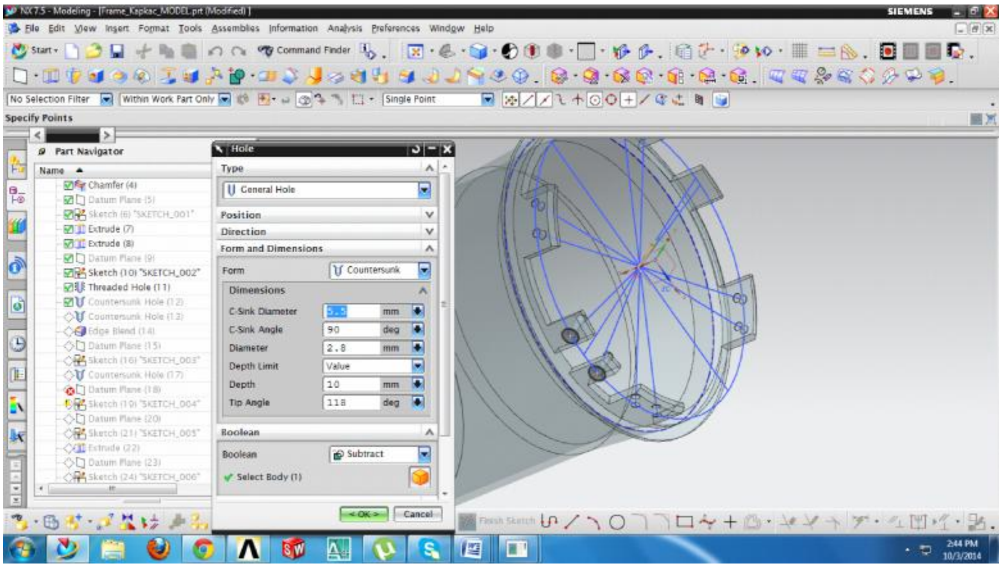
Below image shows the sketch of the Airframe Missile Component.

Below image shows extrude option.

Below image shows the sketch of the Airframe Missile Component.

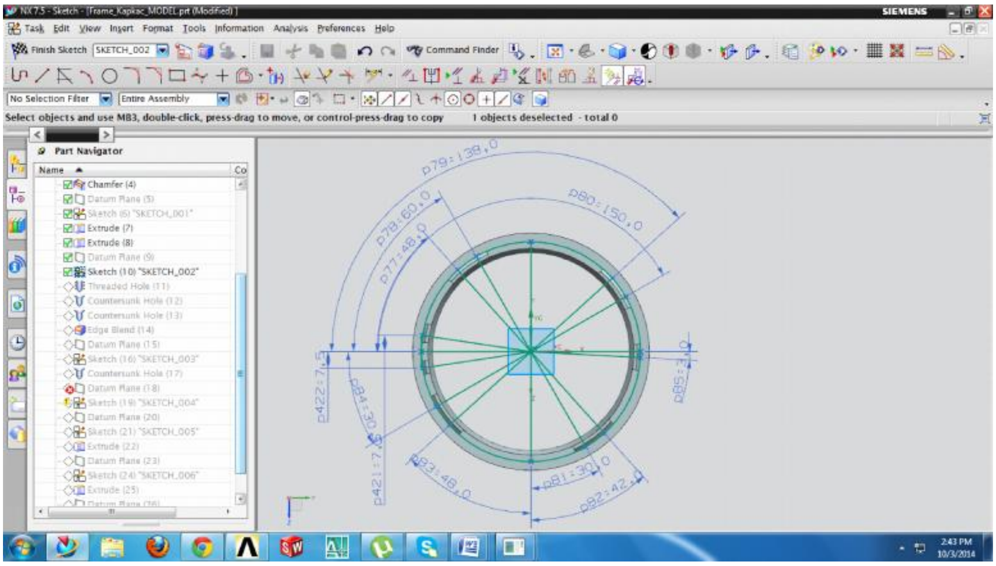
Below image shows the hole option.

Figure 10: Hole



Note: Generate holes on the body by using Simple, Counter bore and Counter sunk.
insert → design features → hole.
In form and dimensions specify type of hole required (countersunk) and dimensions of hole (6dia, 20 depth). In Boolean select subtract option then click ok.

Figure 11: Edge Blend



Note: Creates round edges between faces.
Insert → detail features → edge blend.
Select edge → specify radius → ok

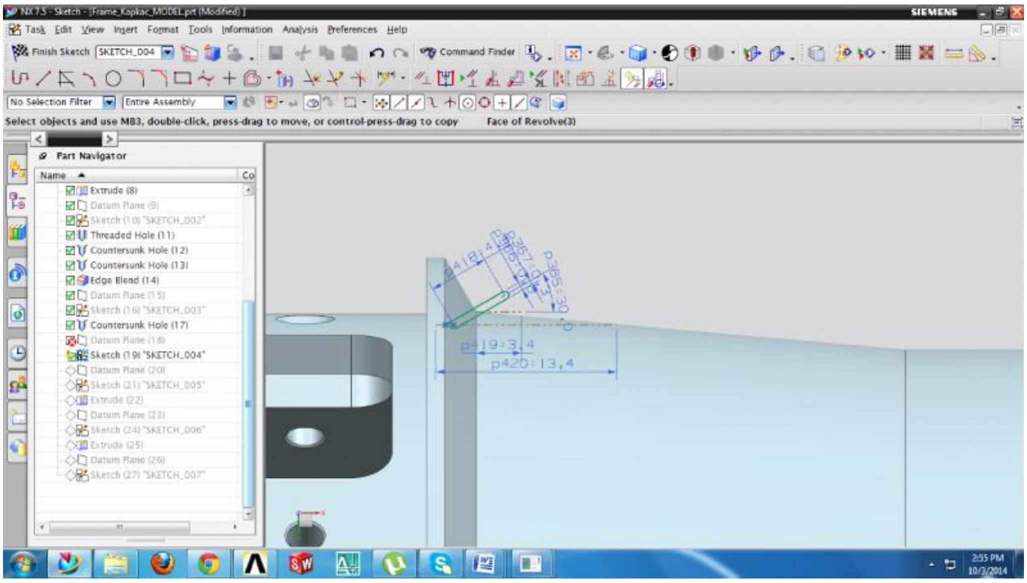
Below image shows edge blend option

Below image shows the sketch of the Airframe Missile Component.

NUMERICAL CONTROL

Numerical Control (NC) refers to the automation of machine tools that are operated

Figure 12



Note: Procedure to draw the above sketch
Insert → sketch in task environment → select plane → ok.
Insert → curve → profile

by abstractly programmed commands encoded on a storage medium, as opposed to controlled manually via hand wheels or levers, or mechanically automated via cams alone. The first NC machines were built in the 1940s and 1950s, based on existing tools that were modified with motors that moved the controls to follow points fed into the system on punched tape. These early servomechanisms were rapidly augmented with analog and digital computers, creating the modern Computer Numerical Control (CNC) machine tools that have revolutionized the machining processes.

In modern CNC systems, end-to-end component design is highly automated using Computer-Aided Design (CAD) and Computer-Aided Manufacturing (CAM) programs. The programs produce a computer file that is interpreted to extract the commands needed to operate a particular machine via a postprocessor, and then loaded into the CNC machines for production. Since any particular component might require the use of a number of different tools-drills, saws, etc., modern machines often combine multiple tools into a single "cell". In other cases, a number of different machines are used with an external controller and human or robotic operators that move the component from machine to machine. In either case, the complex series of steps needed to produce any part is highly automated and produces a part that closely matches the original CAD design.

SELECTION OF SUITABLE MATERIAL

Types of Materials

Aluminum

Brass

Cast iron

Mild steel

Stainless steel

Plastics

Aluminum used as raw material for piston.

Aluminum Specifications

Pure aluminum is alloyed with many other metals to produce a wide range of physical and mechanical properties. The alloying elements are used as the basis to classify aluminum alloys into two categories: non-heat-treatable and heat-treatable.

Most of aluminum specifications designate aluminum alloys in the following way:

- First digit - principal alloying constituent(s),
- Second digit - variations of initial alloy,
- Third and fourth digits - individual alloy variations (number has no significance but is unique).

Cast Aluminum Specifications: The designation system and specifications for cast aluminum alloys are similar in some respects to that of wrought alloys. The cast alloy designation system also has four digits and the first digit specifies the major alloying constituent(s). However, a decimal point is used between the third and fourth digits to make clear that these are designations used to identify alloys in the form of castings (0) or foundry ingot (1, 2).

A letter before the numerical designation indicates a modification of the original alloy or an impurity limit. These serial letters are assigned in alphabetical sequence starting with A, but omitting I, O, Q and X, with X being reserved for experimental alloys. The cast alloy group is shown as: 1xx.x-Pure Al (99.00% or greater), 2xx.x Al-Cu alloys; 3xx.x Al-Si + Cu and/or Mg; 4xx.x Al-Si; 5xx.x Al-Mg; 7xx.x - Al-Zn; 8xx.x- Al-Sn; 9xx.x-Al+Other elements and 6xx.x unused series.

CAM PROCESS

Computer Aided Manufaturing

Computer-Aided Manufacturing (CAM) is the use of computer-based software tools that assist engineers and machinists in manufacturing or prototyping product components and tooling. CAM is a programming tool that makes it possible to manufacture physical models using computer-aided programs.

Manufacturing as the design stage is a set of activities assigned to the producing of the designed part. The manufacturing is one of other activities after design stage. The problem consists in transformation of the CAD data to the manufacturing data. The manufacturing data are sometimes called as CAM data.

The 10 largest CAM software products are:

1. Catia
2. Cimatron
3. Edge cam
4. Master cam
5. NX Cam
6. Power mill
7. Pro/E
8. Space-E/CAM
9. Tebis
10. WorkNC

Selection of Machine

Number of different machines is used with an external controller and human or robotic operators that move the component from machine to machine. In either case, the complex series of steps needed to produce any part is highly automated and produces a part that closely matches the original CAD design. CNC 4-axis turning machine is used for manufacturing of bearing stopper.

Figure 13

RESULTS

The program will be Executing from

Folder: C\Program files\UGS\NX7.5\UGII

Description: NX example with datum CSYS

2D drafting ¬ Drawing in NX ¬ 3D Modeling component using UG NX 7.5 ¬ Datum coordinate system and Sketch -000 using profile curve ¬ Revolve ¬ Chamfer ¬ Skech-001 ¬ Extrude ¬ hole (threaded hole counter shank hole) ¬ Sketch-003 ¬ Edge blend ¬ Part navigation ¬ Sketch-004 ¬ Detailed future ¬ Simulation ¬ Feed rate ¬ Animation speed ¬ Faceted solid ¬ Motion display.

Final Program

N0030 T12 M06

N0040G0G90 X-9995 Y-2105 A62 B270 .5200 M03

N0050 G43 Z2. 5177 H00

N0060 Z2.0454

N0070 G1Z1.9273 F2 M05

N0080 X . 60432

N0090 X . 6028

N00100 X . 4546 F1.5

N00110 X - 3064 F1

N00120 Y - 0.0623

N0130 Y - 0.074 F 1.5

N0140 Y-0.2105 F1

N0150 X - 0.4546

N0160 X - 0. 9995

N0170 X - 0.9995

N0180 Z 2.0454

NO190 G0 Z 2.5197

N0200 Y - 0.2105

N0210 Z 1.9666

N0220 G1 Z 1.966

-

-

N0290 Z 0.2105 Y1

CONCLUSION

The airframe missile is a small light weight booster bearing and infrared homing surface -to-air missile in used primarily as a point defense weapon against Anti-ship cruise missile. The missile is rolls around its longitudinal axis to stabilize it flight path like a bullet fired from a rifled barrel.

This was manufactured by using PLM software of UGNX 7.5 CAD and the missile shield manufacturing involves 2D drafting, 3D design modeling, CAM generation, and simulation. Initially drafting can be done by using GD and T symbols and drawing in NX with all representations. Then the 3D modeling was designed by using CAD NX7.5 software. By using revolving command we can convert 2D modeling to 3D modeling only for axis symmetric bodies the extrude command is used to create a body by sweeping a 2D or 3D section of cures edge sketch in specified direction.

The output from the CAM software is usually a simplex text file of G-codes referred to DNC program with supporting software and the CNC referred to end-to-end computer design automated CAD/CAM program. The Simulation involves the tool path visualization. Finally the tool path listing has 67 lines with G-codes.

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