1. <u>STUDY OF CAD SOFTWARES</u>
a. <u>STUDY OF IRONCAD</u>

<u>AIM</u>

To study about IRONCAD.

<u>About IRONCAD</u>

IRONCAD is a cost effective, full-featured, detail design production tool for engineering design users. It utilizes a new Innovative Design approach, which allows greater design flexibility and capability, resulting in better products in a shorter time. **IRONCAD** provides an alternative to the current history based parametric systems by utilizing a radical drag-and-drop 3D environment with Direct Face Modeling, allowing unmatched ease-of-use and the ability to generate fully associative manufacturing drawings.

<u>System Requirements</u>

• Microsoft Windows 95, 98, and Windows NT 4.0 (Windows 2000 is not currently supported)

• Pentium class or greater processor

• CD-ROM drive

• 64 MB or more preferred

• VGA (256 color) or better, supported by Windows NT, 95, 98 or greater

• Support of OpenGL accelerated graphic hardware

• Typical install 95 MB of hard disk space

Creative Design Approach

The power of Creative Design is in the freedom it gives the user to choose when and where to apply a traditional parametric approach. This freedom of choice allows the user to apply the best design approach for the job at hand. In the early stages of a design it is hard to know exactly what constraint scheme would work the best. With IronCAD the user does not have to concern themselves about which constraint scheme to adopt instead they can focus on the conceptual design issues leaving the parameterization until later in the design process.

Creative Design Benefits

The Creative Design approach delivers the following key benefits to the user.

- **Improved Design Productivity:**

  - **Drag and Drop** – Drag and Drop of basic geometry features from pre built catalogs allows models to be built faster than with the current CAD approach of first building a profile, and then performing some operation on it.

  - **In context design** - Traditional CAD systems create what is called a feature tree that captures the order in which the features were combined to create themodel.When you want to edit a feature in this tree all the features that occur after that point in the history are blanked out.

  - Software that utilizes this approach creates a non-productive situation for the user because many times the feature they would like to reference has been removed. IronCAD does not have this restriction. A user can reference any visible feature on a part while making any required changes. This makes it both easier and faster for the user.

  - **Single design environment** – IronCAD is very unique in that it does not require the support of a separate assembly and part file, instead it only requires a single file which has the ability to support both parts and assemblies seamlessly.

- **Catalogs** - Catalogs provide a fast method to quickly apply design components such as Intellishape features, parts, assemblies, textures, animations, etc, to the model by simply using the Windows drag and drop standard interface

- **Design Flexibility**

  - **Import/Export** - IronCAD has a complete set of data translators that allow both facet and solid geometry to interchange with all leading systems. IronCAD allows engineers to confidently exchange and make use of CAD data throughout the design process, as well as enabling power users and novices to share parts or assemblies.
  - **Dual Kernel** – IronCAD supports both the ACIS and Parasolid kernels simultaneously allowing the user to seamlessly switch between the two. Through kernel collaboration if one kernel fails to perform a certain operation the other kernel is automatically called to see if it has better success.
  - **DFM** – Direct Face Modeling (DFM) allows model modifications to be made irrespective of the previous feature history or in cases where it does not even exist. DFM is very effective when working with imported b-rep geometry that does not have feature history.

- **Ease of Use**

  - **Handles** – IronCAD was a pioneer in the use of dynamic handles for mainstream mechanical design. From its earliest beginnings this patented approach has consistently been enhanced until today it is the most productive way to design available anywhere.
  - **Smart Snap** – Smart Snap, a dynamic 3D constraint solver, which provides precise positioning and alignment while graphically dragging Intellishape features. It allows the user to see which face, edge, and vertex or center point they are about to select for the active command.

- **TriBall** – The patented TriBall, a remarkably intuitive multi-purpose tool provides precise positioning or copying of any objects by translation, rotation, mirroring, patterning or any complex 3D transformation.

- ## Improved Design Visualization and Communication

  - **Rendering** - The integration of powerful, high-end visualization technologies into the core of the software makes the entire design process faster, more productive, and more understandable. Fully integrated, real-time, texture, bump, and decal mapping to represent design details are available to save time and valuable data space.
  - **Animation** – Through the use of Smart Motions IronCAD parts and assemblies can be given realistic motion paths allowing the visualization of dynamic designs.

- ## Production Drawing Automation

  - **Fully associative** – IronCAD supports bi-directional associativity between the 3D model and its 2D drawing. Changes made in the scene can be automatically updated in the drawing and changes made in the drawing can be updated back in the 3D model
  - **Detail drawing generation** – Associative 2D multi-sheet layout drawings can be created automatically from parts and assemblies.
  - **Bi-directional BOM** – BOM creation is made simple through the automated extraction of designated part properties into a fully editable table on the drawing.

RESULT

Thus the various components and features of IRONCAD are studied.

b. <u>STUDY OF NISA</u>

<u>AIM</u>

To study about NISA family of softwares.

<u>About NISA</u>

NISA is one of the most comprehensive engineering analysis suites available globally to address the Automotive, Aerospace, Energy & Power, Civil, Electronics and Sporting Goods industries. Cranes softwares provides global customers with solutions in the areas of Stress Analysis, Seismic Analysis, Vibration Analysis, Composite Material Analysis, Motion & Linkage Analysis, Fatigue Analysis, Thermal Analysis, PCB Analysis, Computational Fluid Dynamics, Electromagnetic Analysis and Civil Structure Analysis. The NISA family includes the following softwares.

<u>NISA/DISPLAY</u>

DISPLAY, a powerful interactive graphics pre and post-processor, makes complex finite element modeling, analysis and results interpretation a cinch.Since its introduction in 1974, DISPLAY has been continuously developed to reflect the latest advances in computer graphics. DISPLAY is required when using any analysis module in the NISA family of programs.

With DISPLAY, 3D graphics have been elevated to a higher level using Open GL capability, which enhances dynamic viewing, shading, lighting and animation capabilities. These have been incorporated for all graphical entities.

<u>NISA Static/Dynamic</u>

NISA is the parent program of the NISA family. It offers an impressive list of linear/non-linear static/dynamic analysis features complemented with an extensive finite element library. NISA has efficient solvers like direct (frontal and sparse) and iterative solvers.

NISA/ROTOR

NISA/ROTOR offers CAE based solutions to a wide variety of problems encountered in the Dynamic Analysis of Rotor-Bearing systems. NISA/ROTOR presents an elegant platform for analyzing different rotor bearing systems frequently encountered in practice. Bearings influence the dynamic behavior of the rotor in a significant way.

Users have been given the facility to input bearing dynamic coefficients using three different options. First one is defining the bearing coefficients directly. Second option is to provide a table of non-dimensional bearing dynamic coefficients vs. Sommerfeld number. From this table, the bearing coefficients at a particular speed are obtained by linear interpolation. Finally, there is a third option to compute bearing coefficients for a specified bearing geometry.

An independent program is incorporated into NISA/ROTOR module to compute the bearing dynamic coefficients. The program utilizes linearized Reynold's equation about a steady state position.

NISA/ COMPOSITE

NISA /COMPOSITES is the most powerful and economical general purpose finite element program to solve a wide range of static, dynamic, buckling, heat transfer, optimization and nonlinear (both geometric and material) analysis problems encountered in Aerospace, Mechanical and Civil Engineering environments.

NISA/3D CAD

To facilitate the integration of analysis with CAD, NISA/3D CAD offers a slew of data exchange capabilities. They take one of the two forms: (1) Direct CAD system interfaces, implemented by a menu command within a CAD program, and (2) Indirect data exchange interfaces, provided by various data translators or built-in commands of DISPLAY Data conversion takes place in a manner transparent to the user.

NISA/HEAT

NISA-Heat is a general purpose finite element program to analyze a wide spectrum of problems encountered in heat transfer.

NISA/ENDURE

NISA-ENDURE is a general purpose software for analyzing the fatigue performance and fracture characteristics of engineering structures and components. The program is completely menu-driven and extremely user friendly. It employs the latest theories to determine crack initiation and crack propagation lives for fatigue analysis and to evaluate fracture parameters. ENDURE provides various analysis models and allows different types of load and material data descriptions such as those encountered in aerospace, offshore, and automobile structures. ENDURE is directly interfaced with NISA II and other commercially available finite element.

NISA/3D FLUID

NISA/3D-FLUID is a general purpose finite element-based, computational fluid dynamics computer code for analyzing fluid flow and heat transfer problems.

NISA/FEAP

FEAP is the largest and most sophisticated finite element program for the stress, random vibration, fatigue life, 3D convective fluid flow and thermal analysis of printed circuit boards (PCBs) and electronic systems.

NISA/OPT

NISAOPT is the most sophisticated family of computer programs available in the world for optimum structural design. The three programs in the NISAOPT family: STROPT, SHAPE, and SECOPT, are fully integrated with NISA II and DISPLAY. State-of-the-art methods are utilized at initial design, intermediate design, and optimum (final) design. Design sensitivity coefficients are also available.

NISA/EMAG

EMAG is the electromagnetic module of the family of general purpose finite element based program NISA. It can determine electric and magnetic field distributions in a wide class of electromagnetic devices. EMAG performs electromagnetic analyses by solving the appropriate Maxwell's formulations. It is coupled with NISA/HEAT to enable a coupled analysis of electromagnetic and thermal fields.

NISA/DYMES

A general purpose program for static, kinematic, and dynamic analyses of constrained multibody mechanical systems undergoing large, nonlinear, three-dimensional displacements. DYMES/Control has been integrated with DYMES for controlled mechanical system simulations.

NISA/CIVIL

NISA/CIVIL offers CAD based solutions to a wide variety of problems encountered in the Analysis and Design of Reinforced Concrete and Steel Structures. Backed by powerful NISA II Analysis and DISPLAY the graphical Pre and Post processor of NISA family of programs, NISA/CIVIL provides excellent tools for modeling, associating design information and carry out design process in Limit state and working stress methodologies of design. Design results are processed to produce structural engineering drawings in AutoCAD environment.

RESULT

Thus the various softwares in NISA family are studied.

c. <u>STUDY OF UNI-GRAPHICS</u>

<u>AIM</u>

To study about Uni-Graphics and its various features.

<u>INTRODUCTION</u>

With a comprehensive range of computer aided design (CAD) applications, NX is unmatched in power and flexibility when it comes to mechanical design and engineering. NX offers a broad set of CAD solutions for the design of complex mechanical products – including dedicated solutions for processes like routed system design, sheet metal design, and plastic part design – that deliver higher efficiency and shorter design cycles at lower costs. NX enables design professionals of any skill level to consider more alternatives, evaluate them more thoroughly, and get to market with innovative designs of superior quality.

NX design applications include tools for:

- Human Modeling
- Modeling
- Assembly Design
- System-based Modeling
- User-defined Features
- Routed Systems Design
- Drafting
- Sheet Metal Design
- Flexible Printed Circuit Design

With NX, efficiency and cost savings extend far beyond the design process to all phases of product development. NX dynamically integrates CAD design with planning, simulation, manufacturing and other development processes to help ensure that design decisions can be made more quickly, with detailed knowledge of product performance and any potential manufacturability issues.

Productive

Complex products and processes demand user interactions and workflows that reduce complexity. NX streamlines decision-making by providing critical information where it is needed. Intelligent user interfaces provide dynamic feedback that accelerates design workflows, whether you are interactively creating product models or weighing performance and manufacturing cost trade-offs.

NX design is intelligent – the productive environment organizes tools, commands and information in straightforward workflows that direct the designer's input to accomplish tasks very efficiently. On-screen displays provide instant feedback and convenient user input control without diverting attention from the task at hand. Part and assembly navigation aids enable designers to quickly comprehend the structure and techniques used to create product models. To reduce of error, NX provides previews for design commands, helping designers to proceed with confidence.

Structured

NX goes far beyond traditional CAD. Using process automation and knowledge capture tools; NX enables reusable and disciplined processes when required. This directly supports processes like lean design, design for six sigma, component and process reuse, and adherence to organizational and industry standards. NX enables companies to balance structured, repeatable best practices with the flexibility they need to move innovation forward.

NX captures design intent through intelligent user input methods, enabling intelligent design modifications and model updates. It leads the designer through an intuitive process of querying ambiguous design decisions that need to be made when changes occur.

Managed

The seamless integration of NX design with data and process management capabilities ensures a continuously up-to-date and synchronized digital product model for

the entire team to use. The NX Engineering Process Management manages and protects your product data with access and security controls. It also helps manage product complexity with automated bill of materials management and synchronization, product configurations, options and variants.

Intuitive workflows in NX present tools to the designer when they are most needed, with real-time feedback.

More UGS CAD/CAM Solutions

- NX CAM Bypasses CAD Step to Streamline Design and Manufacture CAD Visualization Tools Improve Assembly Accuracy (Design Product News CAD Data Isn't Just For Engineering France's Second Largest CAD/CAM/ERP Supplier Declares UGS' Parasolid World's Best 3D Geometric Modeling Software after Exhaustive Benchmark

Overview

- Industrial Design & Styling (CAID)
  - o Freeform Shape Modeling
  - o Visualization
  - o Automotive Styling
  - o Reverse Engineering
  - o Engineering Integration
- Design (CAD)
  - o Modeling
  - o Human Modeling
  - o Assembly Design
  - o System-based Modeling
  - o User-defined Features
  - o Routed Systems Design
  - o Sheet Metal Design
  - o Drafting
- Simulation (CAE)

- NX Nastran
- I-deas NX Simulation
- NX Optimization Wizard
- NX Strength Wizard
- NX Design Validation
  - NX Quick Check
- Tooling
  - NX Mold Design
  - NX Progressive Die Design
  - NX Automotive Stamping Die Design
  - NX Electrode Design
  - NX Jig and Fixture Design Tools
  - NX Advanced Tooling Solution
- CAM
  - Overview
  - Turning
  - 3 Axis Machining
  - High Speed Machining
  - 5 Axis Machining
  - Multi-function Machining
  - Wire EDM
  - Machining Simulation
  - Programming Automation
  - Post Processing & Post Library
  - NC Data Management
  - Part Planning
  - Shop Documentation
  - Resource Management
  - Data Exchange
  - CAD for NC Programming
- Engineering Process Management
- Programming & Customization

- o Knowledge-based Programming
- o Journaling
- o APIs
- o Customization
- I-deas NX Series

- NX Press Releases
- NX Case Studies & Videos
- NX Collateral Library

- Already own NX?
- NX Beta Testing Program

- Contact UGS about NX Software
- Find a UGS Channel Partner

CAM

NX CAM delivers a complete and proven solution for machine tool programming that enables companies to maximize the throughput of their most advanced machine tools. With NX CAM, companies can transform their NC programming and machining processes to dramatically reduce waste and significantly boost productivity.

Optimizing I-deas NX

To smoothen the value of transition from the latest machine tool equipment

NX CAM offers you advanced programming technology developed over many years of experience enabling youold SDRC I-DEAS to get the most from your advanced machine. For High speed machining, 5-axis machining, Multi-Function Machiningnew NX CAM offers the range of flexible and advanced processors needed for effective programming.

All in one system

In NX CAM you have all the pieces you need for, UGS sell a complete NC programming system. It provides a full set of toolpath processors (drilling, turning, 2-5

axis milling, Wire EDM, feature based machining), post processor build and edit tool, full 3D machine tool simulation and a market leading CAD technology for part, tool and machine modeling & assembly.

Programming Automation

UGS sees the value of automation as a means of achieving a competitive business advantage in NC programming. NX CAM offers out of the box capabilities ranging from Process Wizards and templates to automated feature based machining capabilities.

Scalable from stand alone CAM to a PLM solution

NX CAM can be purchased as a stand alone NC programming system, as a CAD/CAM solution, or with CAM data management and tool library solutions. It can be implemented along side NX tool design applications to provide a unified solution. NX CAM can be a key component of an entire part manufacturing solution including part planning, shop floor connectivity and more.

Top 10 reasons to use NX CAM

1. **Advanced capability** in key machining areas including High Speed Machining, 5-Axis machining and the support of multi-function machine tools such as mill-turns, maximizes the value of your advanced equipment.
2. **In depth functionality** that gets the programming job done, even on the toughest parts.
3. **Complete programming package** with everything from a wide range of NC programming functions in the same system: turning, basic milling and drilling, fixed axis milling, variable axis milling, feature based automated programming, support for multi-function machining and wire EDM. You only need one CAM system.
4. **Ease of use.** With a windows based user interface, NX CAM has a familiar look & feel. Menus with graphical content make the system easy to learn and use.

5. **Integrated supporting functions**. NX comes with translators, integrated machining simulation, a graphical post processor editor and a shop documentation generator. - No need for expensive add-on solutions from different vendors.

6. **Available with full CAD functions**, everything you need from drafting, feature based modeling, surface modeling to assemblies. Allows the programmer to handle all geometry issues.

7. **Effective automation tools** makes the task faster, easier to repeat, easier to learn and to use and ensures the use of your preferred manufacturing methods and resources.

8. **Proven with thousands of customers**, from many of the world's largest companies to thousands of typical machine shops of all sizes and in all areas of manufacturing from aerospace to medical devices. UGS understands your needs.

9. **World class technical support** through UGS Global Technical Access Center (GTAC).

10. **CAM data management**, extensions for tool libraries, resource management, part process planning.

RESULT

Thus the various features and applications of Uni-Graphics are studied.

d. STUDY ABOUT ANSYS

AIM

To study about ANSYS and its various features.

INTRODUCTION

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

ANSYS is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering, as well as the Physics and Chemistry departments ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behavior of the product, be it electromagnetic, thermal, mechanical etc.

Generic Steps to Solving any Problem in ANSYS

Like solving any problem analytically, you need to define (1) your solution domain, (2) the physical model, (3) boundary conditions and (4) the

physical properties. You then solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too complex situation. Below describes the processes in terminology slightly more attune to the software.

Build Geometry

Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinate system within ANSYS.

Define Material Properties

Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

Generate Mesh

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces.

Apply Loads

Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions.

Obtain Solution

This is actually a step, because ANSYS needs to understand within what state (steady state, transient? etc.) the problem must be solved.

Present the Results

After the solution has been obtained, there are many ways to present ANSYS. Results choose from many options such as tables, graphs, and contour plots.

Specific capabilities of ANSYS

Structural

Structural analysis is probably the most common application of the finite element method as it implies bridges and buildings, naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

☐ **Static Analysis** - Used to determine displacements, stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

☐ **Transient Dynamic Analysis** - Used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.

☐ **Buckling Analysis** - Used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigen value) buckling and nonlinear buckling analyses are possible.

In addition to the above analysis types, several special-purpose features are available such as **Fracture mechanics**, **Composite material analysis**, **Fatigue**, and both **p-Method and Beam analyses.**

Thermal

ANSYS is capable of both steady state and transient analysis of any solid with thermal boundary conditions.

Steady-state thermal analyses calculate the effects of steady thermal loads on a system or component. Users often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. A steady-

state analysis also can be the last step of a transient thermal analysis; performed after all transient effects have diminished. ANSYS can be used to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convection
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material vary with temperature. This temperature dependency being appreciable, the analysis becomes nonlinear. Radiation boundary conditions also make the analysis nonlinear. Transient calculations are time dependent and ANSYS can both solve distributions as well as create video for time incremental displays of models.

Fluid Flow

The ANSYS/FLOTRAN CFD (Computational Fluid Dynamics) offers comprehensive tools for analyzing two-dimensional and three-dimensional fluid flow fields. ANSYS is capable of modeling a vast range of analysis types such as: airfoils for pressure analysis of airplane wings (lift and drag), flow in supersonic nozzles, and complex, three-dimensional flow patterns in a pipe bend. In addition, ANSYS/FLOTRAN could be used to perform tasks including:

- calculating the gas pressure and temperature distributions in an engine exhaust manifold
- studying the thermal stratification and breakup in piping systems

☐　　using flow mixing studies to evaluate potential for thermal shock

☐　　doing natural convection analyses to evaluate the thermal performance of chips in electronic enclosures

☐　　conducting heat exchanger studies involving different fluids separated by solid regions

FLOTRAN analysis provides an accurate way to calculate the effects of fluid flows in complex solids without having to use the typical heat transfer analogy of heat flux as fluid flow. Types of FLOTRAN analysis that ANSYS is able to perform include:

☐　　Laminar or Turbulent Flows

☐　　Thermal Fluid Analysis

☐　　Adiabatic Conditions

☐　　Free surface Flow

☐　　Compressible or incompressible Flows

☐　　Newtonian or Non-Newtonian Fluids

?　　Multiple species transport

Acoustics is the study of the generation, propagation, absorption, and reflection of pressure waves in a fluid medium. Applications for acoustics include the following:

☐　　Sonar - the acoustic counterpart of radar

☐　　 Design of concert halls, where an even distribution of sound pressure is desired

☐　　Noise minimization in machine shops

☐　　Noise cancellation in automobiles

☐　　Underwater acoustics

☐　　 Design of speakers, speaker housings, acoustic filters, mufflers, and many other similar devices.

☐　　 Geophysical exploration

Within ANSYS, an acoustic analysis usually involves modeling a fluid medium and the surrounding structure. Characteristics in question include **pressure distribution** in the fluid at different frequencies, pressure gradient, **particle velocity**, the sound pressure level, as well as, scattering, **diffraction**, transmission, **radiation**, **attenuation**, and **dispersion** of acoustic waves. A coupled acoustic analysis takes the fluid-structure interaction into account. An uncoupled acoustic analysis models only the fluid and ignores any fluid-structure interaction.

The ANSYS program assumes that the fluid is compressible, but allows only relatively small pressure changes with respect to the mean pressure. Also, the fluid is assumed to be non-flowing and inviscid (that is, viscosity causes no dissipative effects). Uniform mean density and mean pressure are assumed, with the pressure solution being the deviation from the mean pressure, not the absolute pressure.

Coupled Field

A coupled-field analysis is an analysis that takes into account the interaction (coupling) between two or more disciplines (fields) of engineering. A piezoelectric analysis, for example, handles the interaction between the structural and electric fields: it solves for the voltage distribution due to applied displacements, or vice versa. Other examples of coupled-field analysis are thermal-stress analysis, thermal-electric analysis, and fluid-structure analysis.

Some of the applications in which coupled-field analysis may be required are pressure vessels (thermal-stress analysis), fluid flow constrictions (fluid-structure analysis), induction heating (magnetic-thermal analysis), ultrasonic transducers (piezoelectric analysis), magnetic forming (magneto-structural analysis), and micro-electro mechanical systems (MEMS).

RESULT

Thus the study of ANSYS and its various features are studied.

e. <u>STUDY OF AMPS</u>

<u>AIM</u>

To study about AMPS and its various features.

<u>INTRODUCTION</u>

AMPS Technologies Company develops the Advanced Multi-Physics Simulation (AMPS) software with the mission of delivering an advanced simulation system with the most up-to-date FE technology and the best computer software engineering. In the past decade, we have developed AMPS from scratch based on the strictest software-engineering principles and the advanced modular component software design.

Started in 1994 with development funds from Ingersoll-Rand and Owens-Corning R&D Centers, we focused on developing an innovative finite element analysis tool for rotor dynamics and glass fiber forming simulations. Since 2003, they have formed AMPS Technologies Company with main goal to deliver office-tool style finite element software based on modern software engineering design and the newest finite element theory.

AMPS Technologies is a dynamic, growing company with a commitment to continual improvement and quality customer support. We pride in delivering the most convenient desktop FEA tools based on creative software engineering, and constantly push the most advanced computational theories to new limits for our customers.

<u>AMPSolid</u>

AMPSolid, the solid modeling front-end, is built on the industrial standard ACIS solid modeling kernel including full model creation / modification / featuring / de-featuring abilities designed specifically for FE simulation. Full model creation history recording/retrieval and forward/backward rolling to ease model creation. Automatic meshing ability is at the press of a button, with advanced options to control 2D/Shell triangle, quad, mixed meshing, or in 3D tetra or hexa meshing. Proprietary FlexMesh hex meshing technology provides another option for a structured hexa/brick meshing directly

from a solid model. The post-processing part includes animation, contouring, vector flow field, user generated arithmetic results combination, volume/area/side result integral inquiry, and all standard reporting features

PROGRAMS

AMPS offers 3 different programs and 2 add-ons, one that is sure to serve your needs and meet your software budget!

AMPSolid: Full Solid Modeling geometry creation and automatic 2d/3d/shell tri/quad/mixed, tetra, hexahedron mesh generator.

AMPView: Pre-and Post-Processor for finite element analysis, material and boundary condition controls. User supplied DLL for special material and boundary conditions.

AMPSol: General-purpose linear AMPS including stress static and transient dynamics, modal, buckling/instability, h-/p-adaptive refinement. Thermal heat transfer analysis with full stress coupling, and basic nonlinear feature.

AMPS Advanced: Expand the AMPS ability with full functions of large strain/rotation deformation, material nonlinearity and full runtime property dependency, advanced contact and arc-length automatic loading stepping and advanced features not included in the linear AMPS module.

AMPS Explicit: Explicit add-on transient stress/thermal module for transient impact/crash/
fracture analysis with h-adaptive refinement.

<u>AMPSol</u>

AMPSol is a multi-physics finite element analysis program developed for simulations in stress, thermal, flow, and electro-magneto analyses, including steady-state, transient/dynamic, instability buckling, and modal analysis abilities with fully coupled formulation. It follows the latest and rigorous FE formulations including large deformation/rotation Lagrangian formulation (TL or UL), weighted Optimal Least-

Squares fluid formulation with arbitrary Lagrangian-Eulerian extension, automatic load stepping/increment with arc-length post-buckling/instability options, and advanced automatic load-balanced multi-thread controls designed for multi-core/multi-processor workstation. AMPS solver options include iterative element-by-element solver, fast direct sparse and the large capacity iterative PCGLSS solver capable of solving millions of equations in minutes.

AMPView

AMPView is a graphical environment for the user to define all necessary finite element modeling parameters such as material data, boundary conditions, and analysis controls. It links the solid modeling geometry to the discrete FE based modeling specifically designed for FE analysis data graphical association. It controls the AMPSol FE analysis, and then retrieves the results in real-time for user inquiries and examinations. The advanced animation method allowing fast interactive examination of the model behavior, with result inquiry functions such as area/line integral inquiry, user created results type, and advanced cutting

Fluid Structure Interaction with ALE Method

AMPS Products are true general-purpose multiphysics packages with these main features:

- The latest FE technology uses the best computer science software engineering implementation. Fully coupled fluid solid interaction with Conservative Weighted Optimal Least-Squares fluid formulation with proven theoretical convergence without any numerical treatment or relaxation. Arbitrary Lagrangian-Eulerian (ALE) for large motion FSI analysis. Stress, thermal, and flow variables are fully coupled with the electro-magneto fields especially suitable for multi-physics electronic, MEM, VLSI design analysis.
- Robust large deformation/finite rotation formulation matching field results and experiments. Rigorous continuum mechanics theory with the optimized numerical implementation for practical applications.

- Flexible and open access Model DataBase (MDB) with toolkit for user extensible development. Advanced and flexible Dynamic Link Library (DLL) user supplied constitutive model for special user material model research and development. Special user interface loading for special user programmed load/flux data.

- Clear, intuitive, easy to follow industrial standardized Windows menu and GUI convention. Online and context sensitive documentation are available at the click of a button. This allows the user to learn the use of AMPS through browsing the menus instead of reading voluminous documentation.

- Full featured solid model creation/modification/feature/de-feature capabilities specialized for FE simulation applications. Standard ACIS solid model geometry data exchange with most CAD products. Automatic tetra/hexa/quad/tri/mixed meshing with full detailed control. Proprietary FlexMesh structured hexahedron meshing for high quality hex meshing.

- Automatic time/step loading control with special arc-length post-buckling controls. It takes the burden of nonlinear analysis off the analyst's shoulders. Fully h-/p-adaptive ability with based on the energy error control and refinement.

- Extensive real-time result animations, contours, vector plots, movie creation designed by the engineers who have used them to examine, understand, and explore intuitive design variations.

- Extensive on-line reference and Getting Started documentation with tutorials, and the full version of AMPS package can be downloaded for free trial for registered users.

Designed specifically for Windows platform, AMPS delivers the best FE features with the fully coupled stress, thermal, fluid flow, electro and magneto abilities with the PC platform pricing. Wide application profiles with customers from corporate R&D, university research, and industrial design engineers in electronics, civil, mechanical,

AMPSol

AMPSol is a multi-physics finite element analysis program developed for simulations in stress, thermal, flow, and electro-magneto analyses, including steady-state,

transient/dynamic, instability buckling, and modal analysis abilities with fully coupled formulation. It follows the latest and rigorous FE formulations including large deformation/rotation Lagrangian formulation (TL or UL), weighted Optimal Least-Squares fluid formulation with arbitrary Lagrangian-Eulerian extension, automatic load stepping/increment with arc-length post-buckling/instability options, and advanced automatic load-balanced multi-thread controls designed for multi-core/multi-processor workstation. AMPS solver options include iterative element-by-element solver, fast direct sparse and the large capacity iterative PCGLSS solver capable of solving millions of equations in minutes. The rich material library includes incompressible Mooney-Rivlin incompressible material, anisotropic thermal/elastic, visco-elastic-plastic model with ORNL extension. The element library contains advanced beam/rod/shell/.brick/tetra/pyramid elements with user extendable material model and programmable user load/flux function with access to all solution variables.

Technology

With the emphasis on realistically coupled multi-physics simulations, many legacy simulation codes patch up segregated components with batch iterative methods to obtain weakly-coupled solutions. No mathematical proof of the solution exists for such an ad hoc method, and the solution convergence is no problem, AMPS formulated these physics in a single element and directly coupled them by using a strain rate based continuum formulation and a consistent least-square based method for all governing differential equations. All variables such as temperature, deformation, flow, electrical potential, stress, time, etc. are directly coupled and can be applied to the same analysis domain by simply clicking a mouse button to activate the desired physics. The consistent formulation correctly simulates strongly coupled problems such as nonlinear large deformation fluid-solid interaction, Joule heating, conjugate heat transfer, electromagnetic-stress contact, and complex material nonlinear dependency such as MEMS applications. AMPS developers have extensive experience in finite element theoretical development, implementations, and applications experiences in civil, mechanical, automotive, nuclear, aerospace, defense, and general FE applications. The most advanced field-tested FE technologies are implemented on the new object-oriented

C++ language with data structures designed specifically for simulations. The main AMPS FE technologies include: automatic h-/p-adaptive error controls, Lagrangian finite strain and large rotation support, complete elastic-visco-plastic material library, Aribtrary Lagrangian Eulerian (ALE) fluid-structure method, Weighted Optimal Least-Squares fluid formulation, automatic load/stepping controls with arc- length post-buckling method, load-balanced multi-thread implementation for fast parallel processing, and the fastest industrial direct sparse and iterative solver (PCGLSS) with minutes of run time for millions of equations.

RESULT

The study of AMPS and its various features is done.

f. EXPERT SYSTEM OF MACHINE ELEMENTS

AIM

To study about expert system of machine elements.

INTRODUCTION

An expert system is a computer program designed to simulate the problem-solving behavior of a human who is an expert in a narrow domain or discipline. An expert system is normally composed of a knowledge base (information, heuristics, etc.), inference engine (analyzes the knowledge base), and the end user interface (accepting inputs, generating outputs). The path that leads to the development of expert systems is different from that of conventional programming techniques. The concepts for expert system development come from the subject domain of artificial intelligence (AI), and require a departure from conventional computing practices and programming techniques. A conventional program consists of an algorithmic process to reach a specific result. An AI program is made up of a knowledge base and a procedure to infer an answer. Expert systems are capable of delivering quantitative information, much of which has been developed through basic and applied research (e.g. economic thresholds, crop development models, pest population models) as well as heuristics to interpret qualitatively derived values, or for use in lieu of quantitative information. Another feature is that these systems can address imprecise and incomplete data through the assignment of confidence values to inputs and conclusions.

One of the most powerful attributes of expert systems is the ability to explain reasoning. Since the system remembers its logical chain of reasoning, a user may ask for an explanation of a recommendation and the system will display the factors it considered in providing a particular recommendation. This attribute enhances user confidence in the recommendation and acceptance of the expert system.

Components of an Expert System

All expert systems are composed of several basic components: a user interface, a database, a knowledge base, and an inference mechanism. Moreover, expert system development usually proceeds through several phases including problem selection, knowledge acquisition, knowledge representation, programming, testing and evaluation.

Knowledge in the Expert System

The knowledge the expert uses to solve a problem must be represented in a fashion that can be used to code into the computer and then be available for decision making by the expert system. There are various formal methods for representing knowledge and usually the characteristics of a particular problem will determine the appropriate representation techniques employed.

Knowledge bases can be represented by production rules. These rules consist of a condition or premise followed by an action or conclusion (IF condition...THEN action). Production rules permit the relationships that makeup the knowledge base to be broken down into manageable units. Having a knowledge base that consists of hundreds or thousands of rules can cause a problem with management and organization of the rules. Organizing rules and visualizing their interconnectedness can be accomplished through dependency networks. This method of knowledge representation is discussed later in this discussion under knowledge acquisition and representation.

During the consultation, the rule base is searched for conditions that can be satisfied by facts supplied by the user. This operation is performed by the inference engine. Once all of the conditions (i.e. IF parts of rules) of a rule are matched, the rule is executed and the appropriate conclusion is drawn. Based upon the conclusions drawn and the facts obtained during consultation, the inference mechanism determines which questions will be asked and in what order. There are various inferencing methods available to perform the tasks of searching, matching, and execution.

29

A distinctive characteristic of expert systems that distinguishes them from conventional programs is their ability to utilize incomplete or incorrect data. Given only a partial data set, an expert is likely to have less than absolute certainty in his conclusion. The degree of certainty can be quantified in relative terms and included in the knowledge base. The certainty values are assigned by the expert during the knowledge acquisition phase of developing the system. By incorporating rules in the knowledge base with different certainty values, the system will be able to offer solutions to problems without a complete set of data. The capacity to deal with uncertainty is available in development software.

Explanation of the Decision Making Process

One of the attractive features of expert systems is the program's ability to review a consultation and provide the user with an explanation for how its conclusion was derived. The explanation function is essentially a record of the reasoning process used by the expert to resolve the problem. It provides for a better understanding of how the conclusion was reached and instills in the user a greater confidence in the conclusion and the expert system. The accumulation of facts to be presented when an explanation is asked for is usually part of the development shell or software.

Making the Expert System Easy to Use

Whether or not an expert system achieves success may be determined by the nature of its user interface. This is the part of the expert system that interacts with the user. Even the most powerful expert system will not be applied if it requires too much effort on the part of the user. For this reason, it is important to make the computer as easy for the user to operate as possible. Almost all modern development programs offer the capacity to interact with the system through both text and graphics.

Developing an Expert System

The power of an expert system is derived from the knowledge of the expert. The lack of computer skills should not inhibit anyone from using this tool in implementing their disease management program. There are published procedures that one can follow to develop expert systems, although rigid adherence to these procedures is not necessarily

a prerequisite for a successful development effort. Match your development technique with your own style of thinking and your resources and problem area. Listed below are several examples of where the developers of the Penn State Apple Orchard Consultant (PSAOC) strayed from accepted developmental procedures and still produced a successful expert system which is advising Pennsylvania apple growers.

Selecting the Problem

In this section we will discuss the development of expert systems in three parts:

- Selecting a problem
- Knowledge acquisition and representation
- Evaluation and adoption. We will concentrate our discussion on the plant pathology related aspects of developing expert systems and will give very little time to discussing expert system software tools. In addition, the Penn State Apple Orchard Consultant will be used as an example to further illustrate each of the developmental phases.

The most critical step in developing an expert system is identifying a suitable problem. Criteria to determine the feasibility of developing expert systems and selecting the problem have been discussed previously. Expert systems are best suited to problems that require experience, knowledge, judgment, and complex interactions to arrive at a solution.

One of the first tests to determine if a subject area is suitable for an expert system is whether the solution of the problem requires the knowledge and expertise of a human expert. For many agricultural situations a reasonably knowledgeable person with the correct information such as a production guide or a data base management system available can determine the solution without the help of an expert or an expert system. The abilities inherent in the expert system software may not be necessary to solve the problem.

Problems such as determining days-to-harvest limitations for a pesticide, or calculating infection periods may be better accomplished by using a production manual,

or a conventional computer program such as a spreadsheet or database manager. In addition, in crop areas where little is known about certain diseases the power of an expert system is really not necessary

Cost of Development

Expert systems are expensive to develop. They require resources, expertise and time to build. It should be determined if the proposed expert system can be justified in terms of savings or other benefits it produces. Costs include software, hardware and personnel to do the work. A developer should ask if the problem to be solved justifies the investment of time and money. If current loss due to disease, whether it be due to reduced yield or quality, is great enough, then the successful implementation of an expert system may offset cost of development.

Sometimes there are other benefits in using expert systems which more than justify cost such as: reduced pesticide inputs, enhanced IPM training, effective delivery of IPM strategies not possible by any other delivery method and identification of gaps in knowledge to direct research programs. Expert systems also have value in that they can make expertise available to decision makers in agriculture when expert opinions are not available. An expert system can be available for the grower to use at any time of the day and it could be available to every grower in the state at the same time.

Knowledge Acquisition and Representation

After the problem has been selected, the knowledge acquisition phase of expert system development is begun. The task now is to have the knowledge which the expert uses to solve the problem displayed in a logical fashion so that it can be coded into the computer.

There are many discussions in the literature on the extraction of knowledge from domain experts. Most expert system developers suggest that a knowledge engineer (someone who is trained in the extraction of information from experts) extract the knowledge and design the expert system. In some cases, a knowledge engineer is needed to 'tease apart' highly compiled expert knowledge because the expert is too familiar with

the area to break it into its logical components. There is also software available to help in self knowledge engineering.

In the development of the PSAOC a different method of knowledge acquisition was undertaken. The domain experts were trained in knowledge acquisition and representation. After the domain experts had represented their knowledge in dependency networks (diagrams of the rules), the networks were given to programmers who coded it into the computer. The domain experts in this case were both the source of knowledge and the designers of the expert system. The most obvious advantage of this development method is that it is much less expensive to build expert systems if it is not necessary to support a knowledge engineer for the extraction and representation of knowledge. However, there is a more important advantage in using the domain experts as the knowledge engineers. Only the domain expert knows the extent of his knowledge, how he thinks about problems and what variables are important in the decision process. Rather than the knowledge engineer painfully extracting the information from the expert, the knowledge flows from the expert as he articulates from internal examination how he makes decisions. The domain expert becomes an active participant in the knowledge acquisition and representation process rather than a passive participant who is acted upon. The domain expert becomes more involved in the development of the expert system as he is challenged to represent on paper and then program into the computer, the methods and processes by which he solves problems. In addition, domain experts can better communicate their logic to other domain experts; an important consideration where different domains overlap and there must be agreement among domains. Also, extension specialists who serve as the domain experts are more familiar with how the end users prefer to think about and solve problems. Their experience with the user clientele will help them to format the expert system in a manner that county agents or growers will best understand and utilize

RESULT

The study about expert system of machine elements is done.

2. <u>STUDY OF INTERNET TOOLS</u>

<u>AIM</u>

To study and familiarize different features in internet.

In this session each candidate has to acquire knowledge about the following
I. Familiarization of Electronic Mail
      1. How to create an e-mail id.
      2. To familiarize free e-mail providing websites.
      3. Send and receive e-mail.
      4. Features in e-mail service such as
            a. Deleting, forwarding, replying etc.
            b. Bulk mail option
            c. Folder creation
            d. Briefcase
            e. Settings
            f. Attaching large files

II. Familiarization of worldwide webs
      1. Education sites
      2. News sites
      3. Sites related to placements
      4. LAN & WAN facility
      5. Awareness of Bulletin Board Service (BBS)
      6. Awareness of Forte Free Agent
      7. Awareness of Netscape Communicator

III. Familiarization of different search engines such as
      1. yahoo
      2. google
            - Acquire knowledge about different groups available in different websites
            - Acquire membership in any of the groups related to education
            - Using message boards

IV. FTP
            - Basic idea on FTP
            - Setting of FTP
            - Different types of FTP

V. Familiarization of Chatting
            - Acquire membership in chat rooms
            - Familiarization of Engineering related chat sites

VI. Study on cyber laws

VII. Basic idea on creation of web pages
                - Use of HTML, Use of database program etc.

<u>RESULT</u>

The study about various internet tools is done.

3. <u>STUDY OF MATLAB</u>

<u>AIM</u>

To study about MATLAB and its various components.

<u>MATLAB</u>

MATLAB is a high-performance language for technical computing. It integrates computation, visualization, and programming in an easy-to-use environment where problems and solutions are expressed in familiar mathematical notation. Typical uses include Math and computation Algorithm development Data acquisition Modeling, simulation, and prototyping Data analysis, exploration, and visualization Scientific and engineering graphics Application development, including graphical user interface building.

MATLAB is an interactive system whose basic data element is an array that does not require dimensioning. This allows you to solve many technical computing problems, especially those with matrix and vector formulations, in a fraction of the time it would take to write a program in a scalar non interactive language such as C or Fortran. MATLAB features a family of add-on application-specific solutions called toolboxes. Very important to most users of MATLAB, toolboxes allow you to learn and apply specialized technology. Toolboxes are comprehensive collections of MATLAB functions (M-files) that extend the MATLAB environment to solve particular classes of problems. Areas in which toolboxes are available include signal processing, control systems, neural networks, fuzzy logic, wavelets, simulation, and many others.

<u>The MATLAB System</u>

The MATLAB system consists of five main parts:

(1)Desktop Tools and Development Environment.

This is the set of tools and facilities that help you use MATLAB functions and files. Many of these tools are graphical user interfaces. It includes the MATLAB desktop

and Command Window, a command history, an editor and debugger, and browsers for viewing help, the workspace, files, and the search path.

(2)The MATLAB Mathematical Function Library.

This is a vast collection of computational algorithms ranging from elementary functions, like sum, sine, cosine, and complex arithmetic, to more sophisticated functions like matrix inverse, matrix eigen values, Bessel functions, and fast Fourier transforms.

(3)The MATLAB Language.

This is a high-level matrix/array language with control flow statements, functions, data structures, input/output, and object-oriented programming features. It allows both "programming in the small" to rapidly create quick and dirty throw-away programs, and "programming in the large" to create large and complex application programs.

(4)Graphics.

MATLAB has extensive facilities for displaying vectors and matrices as graphs, as well as annotating and printing these graphs. It includes high-level functions for two-dimensional and three-dimensional data visualization, image processing, animation, and presentation graphics. It also includes low-level functions that allow you to fully customize the appearance of graphics as well as to build complete graphical user interfaces on your MATLAB applications.

(5)The MATLAB External Interfaces/API.

This is a library that allows you to write C and Fortran programs that interact with MATLAB. It includes facilities for calling routines from MATLAB (dynamic linking) calling MATLAB as a computational engine, and for reading and writing MAT-files.

Programming

MATLAB is a high-level language that includes matrix-based data structures, its own internal data types, an extensive catalog of functions, an environment in which to

develop your own functions and scripts, the ability to import and export to many types of data files, object-oriented programming capabilities, and interfaces to external technologies such as COM, Java, programs written in C and Fortran, and serial port devices. The MATLAB programming features and techniques in the following chapters:

- Data Structures
- Data Types
- Basic program components
- M-File programming
- Data Import and export
- Error handling
- Programming Tips

Graphics

The important features of MATLAB in graphics are the following:

(1)MATLAB Plotting Tools: -It helps in creating plots and setting graphic object properties

(2)Data Exploration Tools: - These tools are used to extract information from graphs interactively

(3)Annotating Graphs: - Here we are able to add annotations, axis labels, titles, and legends to graphs.

(4)Basic Plotting Commands: - This command helps in plotting vector and matrix data in 2-D representations

(5)Creating Specialized Plots: - This enables in creating bar graphs, histograms, contour plots, and other specialized plots

(6)Displaying Bit-Mapped Images: - It is used for displaying and modifying bit-

mapped images with MATLAB

(7)Printing and Exporting: - This method is used for printing graphs on paper and exporting graphs to standard graphic file formats

(8)Handle Graphics Objects: -MATLAB graphics objects and properties

(9)Figure Properties: - How to use figure properties

(10)Axes Properties: - How to use axes properties

3-D Visualization

MATLAB also have 3D visualization techniques and the application of these techniques to specific types of data.

- Creating 3-D Graphs
- Defining the View
- Lighting as a Visualization Tool
- Transparency
- Creating 3-D Models with Patches
- Volume Visualization Techniques

RESULT

Thus the various features and components of MATLAB are studied.

4. STUDY AND EXPERIMENT ON MICRO STRUCTURE OF SPECIMEN

AIM

To identify the given specimen by conducting suitable metallurgical examination of the given specimen

APPARATUS REQUIRED

Metallurgical microscope (1:100), polishing machine, emery paper (220, 440, 600, etc), etching material

THEORY

Metallography is the science and art of preparing a metal surface for analysis by grinding, polishing, and etching to reveal microstructual constituents.

Grinding is a most important operation in specimen preparation. During grinding the operator has the opportunity of minimizing mechanical surface damage that must be removed by subsequent polishing operations. Even if sectioning is done in a careless manner, resulting is severe surface damage; the damage can be eliminated by prolonged grinding.

Polishing is the final step in production a surface that is flat, scratch free, and mirror like in appearance. Such a surface is necessary for subsequent accurate metallographic interpretation, both qualitative and quantitative. The polishing technique used should not introduce extraneous structure such as disturbed metal, pitting, dragging out of inclusions, comet tails and staining.

Although certain information may be obtained from as-polished specimens, the microstructure is usually visible only after etching. Only features which exhibit a significant difference in reflectivity (10% or greater) can be viewed without etching. This is true of micro structural features with strong color differences or with large differences in hardness causing relief formation. Cracks, pores, pits, and nonmetallic inclusions may

be observed in the as-polished condition. In most cases, a polished specimen will not exhibit its microstructure because incident light is uniformly reflected. Since small differences in reflectivity cannot be recognized by the human eye, some means of producing image contrast must be employed. Although this has become known as "etching" in metallography, it does not always refer to selective chemical dissolution of various structural features. There are numerous ways of achieving contrast. These methods may classified as optical, electrochemical (chemical), or physical, depending on whether the process alters the surface or leaves if intact.

After preparation, the sample can easily be analyzed using optical or electron microscopy. Prepared samples are most often inspected using an inverted metallographic microscope. This type of microscope is sufficient for magnifications less than 1000x. If sample must be observed at higher magnification, it will be examined in a scanning electron microscope (SEM).

<u>PROCEDURE</u>

The given specimen, the end surface was made flat by filing to get a clean boundary when viewed .Then the specimen is polished by water emery paper of different grades of no.s 220,300,440,600 etc. The no.600 emery is attached on the rotating disc.

During finest polishing process, the Alumina (Al_2O_3) powder was applied between the specimen and rotating cloth covered wheel. Water may be used as lubricant and to remove /flesh out the particles cut from the surface.

In the etching process, the specimen was cleaned with etchent to make the grain boundaries visible. For etching the polished surface is immersed in a weak acidic or alcoholic solution. Suitable solution (etchents) for specimen of various metals is listed. The etched specimen is now ready to be viewed in a metallurgical microscope. Adjust the focus for clear image. Then the structure was compared to photo micrograph and the metal was identified.

Metal Specimen	Name of Etchent	Decomposition	Percent of Concentrate	Etching time(in seconds)
Iron, Steel, Cast iron	Nital	HnO_3,Methale alcohol	2 , 98	20
Stainless Steel		Nitricacid,Hydrochloric acid	88 , 12	300
Aluminum and alloys	Kollers reagent	Hydrochloric acid, HCl HNO_3,Distilled water	1, 1.5 2.5 , 95	15
Copper and Alloys		Ferric chloride, HCl Met hale alcohol	5 , 2 93	100
Zinc and alloys	Nital	HnO3,Methale alcohol	2 , 98	10-300

RESULT

        The metallurgical test was conducted and the specimen was identified as -----------

5. STUDY AND EXPERIMENT ON PROFILE PROJECTOR

AIM

1. To make a study of profile projector
2. To determine the nomenclature of the given screw thread, gear tooth, tool
3. Surface examination for minute cracks and surface irregularities

APPARATUS

The profile projector available in the lab is Metzer optical projector manufactured by Metzer India optical instrument company.

THEORY

The optical projector displays the magnified images of the objects located by some holding device or slaging fixture or an approximate screw.

An optical projector consist of-

1. A projector drawing a light source, condenser or collimating lens system to direct the light part by part into an elliptical system
2. Suitable work holding table
3. Projection optics including mirror and lenses
4. Screen where the image of work piece is projected and where the measurements are made.
5. Measuring device

The image of the tool is produced on the screen by a beam of light. The light source being a tungsten lamb, filament lamb or high pressure mercury lamb. A strong beam of light consisting of bundle of light rays which are parallel is produced by optical means. Beam should be of large enough diameter to provide convergence of tool and accurate illumination on the screen. The common arrangements of light screen used in optical projector are- (i) shadow (ii) surface.

The shadow image of the object may be produced either on horizontal or vertical screen surface illumination produces a reflected image of the face of the object.

The light beam after passing on to the projector system comprising of lenses and mirrors which must be held in accurate alignment on rigid support. The lenses are used to get the desired magnification and mirrors to direct the beam of light on screen. An optically flat glass is set on to the top of the table and glass surface in addition to allow the passage of light beam also serves a good surface positioning of a variety of flat paths.

PROCEDURE

Place the object on the micrometer stage. Switch on the transmitted light source. An image will be obtained on the screen. Do the focusing of image by adjusting the handle provided. Adjust the image within the screen by selecting the magnification factor of the objective scale.

Trace out the image on the tracing paper. Calculate the required dimension from the observed values.

a) For examining the crack

For the examination of the surface irregularities and minute cracks, switch off the transmitted light source and switch on the incident light source. Place the object on the stage for the examination. Adjust the focus and observe cracks. The length and breadth of the cracks can be measured by adjusting the micrometer of the stage.

b) For screw thread and gear tooth

The linear measurements in X and Y directions are done with the help of the micrometer provided. To measure the angle, the image obtained on the screen is made clear by sliding the tube up or down, which is done by rotating the hand wheel. Now the tracing paper is placed on the screen and the image is sketched on to it. The procedure is repeated by altering the position of the tool on the glass surface, so that various angles are obtained.

c) For cutting tool

The cutting tool is placed on the glass surface of the profile projector. The lights on the projector are switched on. The image obtained on the screen is made clear by sliding the tube up or down, which is done by rotating the hand wheel. Now the tracing paper is placed on the screen and the image is sketched on to it. The procedure is repeated by altering the position of the tool on the glass surface, so that various angles are obtained.

RESULTS AND INFERENCES

The study of profile projector is completed.

Tool angles

Front cutting edge angle = _____

Side cutting edge angle = _____

Front clearance angle   = _____

Back clearance angle   = _____

Screw thread

Pitch   =   ------------------

Lead   =   -----------------

Angle   =   -----------------

Gear tooth

Sl no	Dimension part	Initial reading	Final reading	Actual dimension

6. STUDY AND EXPERIMENT ON JOURNAL BEARING APPARATUS

INTRODUCTION

Journal bearing apparatus is designed on the basis of hydrodynamic bearing used in practice.

To formulate the bearing action accurately in mathematical terms is a more complex job. However, one can visualize the pattern of bearing pressure distribution due to the hydrodynamic action with the help of experimental rig. This helps to understand the subject properly.

The experimental rig consists of a small journal bearing as shown in fig. This apparatus helps to demonstrate and study the effect of important variables such as speed, velocity and load on the pressure distribution in a journal bearing. This pressure distribution can be verified with Sommerfield equations.

DESCRIPTION OF APPARATUS

The apparatus is illustrated in Fig. No. 1. It consists of a M.S. bearing mounted freely on a steel journal shaft (A). This journal shaft is fixed directly on to a motor shaft (S). The speed of the PMDC motor is finely controlled by a D.C dimmerstat. The journal bearing (E) has twelve (No.1 to 12) equi-spaced pressure tapping around its circumference and four No. A, B, C, D additional pressure tapping are positioned on the topside of the bearing. The two sides of bearing are closed with two M.S. plates and sealed with gasket packing to avoid leakage. Small balancing weights (J) are provided to maintain the bearing in its normal position during the test run and while taking the readings. Both the weights can be adjusted freely along the rod.

The film pressure is indicated by using a pressure transducer/indicator. For this, tappings are taken from the bearing to common header pipe using valves for each tapping. Thus pressure at each point spaced around the bearing is known by selecting the valve every time. The oil reservoir can be adjusted at required height and connected to the bearing through plastic tube.

EXPERIMENT ON JOURNAL BEARING APPARATUS

AIM

To observe the pressure profile of a journal bearing at various conditions and after noting the pressure profile for any chosen condition, the following analysis are to be conducted.

      a) Plotting the Cartesian and polar pressure curves
      b) Plotting the theoretical and polar curves.

SPECIFICATIONS

Journal diameter:

Bearing diameter:

Bearing width:

Weights: sets of adjustable weights

Recommended oil: Any lubricating oil

Motor: DC, 3000RPM

Control unit: special design to give variable speed

Manometer panel; 16 tubes, 300cm (approx)

THEORY

The Sommerfield pressure function is given by

$$P - p_o = \frac{-6\mu R_2^2 \omega\, \varepsilon \sin\upsilon (2 + \varepsilon \cos\upsilon)}{c^2 (2 + \varepsilon^2)(1 + \varepsilon \cos\upsilon)^2}$$

Where 'p' is the pressure of the oil film at the point measured clockwise from the line common centers and (00') and

$P = P_0$ at $\upsilon = 0$ and $\upsilon = \pi$ (ref figure)

ω=speed of rotation of journal

R_2=radius of journal

R_1=radius of bearing

C=radial clearance

ε=eccentricity

μ=viscosity of oil

$h=(c+ \varepsilon \cos\upsilon)^2$

Some books on lubrication give the Sommerfield function with a negative sign for ε. this is true if υ is measured from the point of minimum thickness of oil film.

$$h=c(1- \varepsilon \cos\upsilon)$$

Also, $\cos\upsilon_m=-3 \varepsilon /(2+ \varepsilon^2)$

Hence the maximum pressure occurs at the point $\upsilon = -\upsilon_m$

The total load P on the journal is given by

$$P= \frac{12\mu R_2^2 L \omega\pi \varepsilon}{C^2(2+ \varepsilon^2) (1+ \varepsilon^2)^{1/2}}$$

which is acting perpendicular to OO'

L= width of the bearing

Total force along OO' is zero.

PROCEDURE

Fill the oil tank by using proper lubricating oil under test and position the tank at the desired height. Drain out the air from all the tubes of the manometer. Check that some oil leakage is there for cooling purpose. Start the motor slowly and reach the desired speed. Select the direction of rotation and switch on the motor. Gradually increase the speed. Let the journal run for about an hour until the oil in the bearing is warmed up. Reduce the speed and allow the bearing to settle down for about 10 minutes. Add the

required loads and return the bearing to vertical position. When the manometer levels have settled down, take the pressure readings on 16 manometers. Repeat the experiment for other speeds. Polar plot of pressure variation may be obtained from the readings on the manometer. After the test is over, bring down the position of the oil tank so that there will be no leakage in the idle period. Change the oil seals when they rupture and leakage starts.

RESULT

Supply head, P_0 = --------- cm

Weight of bearing = -------- kg

Speed (rpm)	Tube no.	P (cm)	P-P$_0$
w_1			
w_2			
w_3			

7. STUDY AND EXPERIMENT ON AUTOCOLLIMATOR

AIM

1. To study the instrument in detail
2. To measure the straightness error of the given surface using autocollimator

STUDY OF INSTRUMENT

The autocollimator is a precision optical instrument for measuring very small angular measurement over a significant distance. They can be used for evaluating alignment of machine surfaces, surface plate flatness, and squareness of one surface to another, straightness of shafts and a variety of other orientation measurement. Interferometers and optical flats are used for some of these evaluations.

The measuring range of autocollimators varies with the design, but they are typically used for angles of under 30 arc minutes. Resolutions of moderately priced instruments are in the 1 arc second angle.

A bright light source is condensed through a lens. The light then goes through a graticule with an etched imaged. This is then projected through a beam splitter, which projects the image on the graticule through the collimating lenses of the autocollimator. The collimated image goes to a precision mirror at the surface to be evaluated. The mirror reflects the image back through the collimating lenses of the interferometer.

If the mirror is precisely perpendicular to the collimated light path, the image will be perfectly aligned with the viewing optic graticule. Otherwise, it will appear as an offset image. The distance (or angle) of deviation can be determined by graduations on the graticule or by adjusting the cross hairs of the graticule to coincide with the image. In the latter type, the amount of movement is tracked by graduations on the adjusting screws that move the cross hairs

The autocollimator is basically a telescope permanently focused at infinity, and incorporating an illuminated target graticule in the focal plane of the lens. Every point on the graticule produces a parallel beam of light from the lens and parallel beams of light entering the lens are brought to focus in the focal plane

The instrument uses one of the basic principles of reflection – that a parallel beam of light projected on to a plane reflector square to the beam, is reflected along the same path. When the reflector is tilted with respect to to the incident beam ,the return angle of the reflected beam is twice the angular displacement of the reflector the amount of this displacement can be read off directly through the instrument eye piece, or by micrometer or by digital display depending upon the type of autocollimator being used .

PROCEDURE

> Find the pitch length of the vertical reflector unit and understand the optical read out system
> Move the vertical reflector along the surface to be tested in steps equal to the pitch length of the vertical mirror.
> Measure angular variation at each position
> Convert these readings into height differences
> Determine the height above and below the starting point
> Convert these heights differences with regard to the datum line that is the line joining the first and the last point, coordinates of the to points being zero.
> Determine the straightness error.

OBSERVATIONS

Position of the reflector	Autocollimat or reading	Rise or fall from first readings (sec)	Rise or fall (micrometer)	Cumulative of readings in column 4	Adjustments to be done to make two end points zero	final rise or fallof each point datum linew.r.t
1	2	3	4	5	6	7
\						

RESULT:

INFERENCE:

8. STUDY AND EXPERIMENT ON CAM PROFILE

AIM

To determine the profile measurement of the given cams, also find the maximum lift.

APPARATUS

V- block, Graduated protractor, Dial test indicator with stand.

THEORY

A cam is a mechanical member for transmitting motion to a follower by direct contact. The cam profile is the actual working contour of the cam. It is along surface of contact of cam and follower. The base circle is the smallest circle drawn to the cam profile from the centre of rotation of the radial cam. Cam angle is the angle of rotation of the cam for the definite displacement of the follower. Dwell period of the cam is the angle of movement of the cam for which the follower has no displacement from the base circle of the cam. Rise of the cam is the angle through which the follower starts to rise and attains maximum displacement or lift. Fall of the cam is the angle through which the follower starts to fall and attains zero displacement. Overlap of two cams is the angle through which both the cam operate the follower simultaneously

PROCEDURE

Place cam shaft on V- block and fix the graduated protractor at the end of the camshaft. Set the dial gauge plunger at dwell position and set dial to zero. Mark the reference point as zero degree. Rotate the shaft gently and note down the deflection at each 10 degree. Repeat the experiment till one complete rotations of camshaft. At the same settings of the dial gauge also note down the deflections of next cam by the same procedure. Draw the base circle of the cam with scale factor and mark the readings. Connect the tabulated points and we will get the profile of the given cams, and also find the lift over lapping period and dwell period.

RESULT

The profile measurement of cam is conducted

1. Maximum valve lift (inlet valve)   = ……………
2. Maximum valve lift (exhaust valve)   = ……………
3. Dwell period   = ……………
4. Overlapping period   =……………..

INFERENCE

OBSERVATIONS

Angle of rotation	Inlet valve deflection (mm)	Exhaust valve deflection (mm)	Angle of rotation	Inlet valve deflection (mm)	Exhaust valve deflection (mm)	Angle of rotation	Inlet valve deflection (mm)	Exhaust valve deflection (mm)
10								
20								
30								
40								
50								
60								
70								
80								
90								
100								
110								
120								

Maximum lift of inlet valve cam   =……………

Maximum lift of exhaust valve cam =……………

Dwell period of cam   =……………

Overlapping period   =…………….