SRINIVASAN ENGINEERING COLLEGE

(Approved by AICTE, New Delhi and affiliated to Anna University, Chennai)

(An ISO 9001:2008 Certified Institution)

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CAD PART

INTRODUCTION

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform Product Life Management (PLM) CAD/CAM/CAE commercial software suite developed by the French company Dassault Systems. Dassault Systems CATIA is mainly developed for Aerospace, Automobile and Manufacturing industries.

INTRODUCTION TO CATIA V5

CATIA V5 is mechanical design software, addressing advanced process centric design requirements of the Aerospace industry. It is the first of the next generation software of Dassault System Software Solutions, France. With its feature based design solutions, CATIA proved to be highly productive of assemblies and drawing generation for aerospace & mechanical industries.

CATIA Version5 is enabled with Multiple Document Interface (MDI). With Window menu, we can manipulate several documents at the same time. CATIA GUI loads with toolbars according to the workbench.

SPECIFICATION TREE

It is a browser containing the history of tools and process used to create the part.

CATIA WORKBENCHES

1. SKETCHER
2. PART DESIGN
3. ASSEMBLY DESIGN
4. DRAFTING
5. GENERATIVE SHAPE DESIGN (WIREFRAME AND SURFACE DESIGN)
6. SHEET METAL DESIGN

TYPES OF SCREEN

1. Working Screen
2. Hide Screen

CONSTRAINT

To arrest the Degrees Of Freedom (DOF) (i.e., to lock the profile from the movement)

CONSTRAINT-TYPES	
COLOUR OF LINE	**Sketch Solving Status**
1.Green	**Iso-Constrained (Fully Constrained)**
2.White	**Under-Constrained**
3. Purple	**Over-Constrained**

CATIA WORKBENCHES

SKETCHER

In general, the design process starts by identifying the fundamental feature of the part and making a sketch for it. CATIA Sketcher tools initially drafts a rough sketch following the shape of the profile.

PART DESIGN

Part Design workbench can be used to convert the sketch into a sketch-based feature. You also provided with other tools to apply the features, such as fillets, chamfers, and so on. These features are called the dress-up features. You can also assign materials to the model in the Part Design workbench. Many mechanical designs consist of complex assemblies made from many parts. This type of design work can be made easier by part and assembly modeling capabilities that are well integrated.

ASSEMBLY DESIGN

The **Assembly Design** workbench is used to assembly the components using the assembly constraints available in this workbench. There are two types of assembly design approaches:

1. Bottom-up
2. top-up

In the bottom-up approach of the assembly, the previously created components are assembled together to maintain their design intent. In the top-down approach, components are created inside the **Assembly Design** workbench. You can also assemble an existing assembly to the current assembly. The **Space Analysis** toolbar provides the **Clash** analysis tool that helps in detecting clash, clearance and contact between components and subassemblies.

DRAFTING

The Drafting workbench is used for the documentation of the parts or assemblies created earlier in the form of drawing views and their detailing. There are two types of drafting techniques:

1. Generative drafting
2. Interactive drafting

The generative drafting technique is used to automatically generate the drawing views of the parts and assemblies. The parametric dimensions added to the component in the **Part Design** workbench during its creation can also be generated and displayed automatically in the drawing views. The generative drafting is bidirectional associative in nature. You can also generate the Bill of Material (BOM) and balloons to the drawing views.

In interactive drafting, you need to create the drawing views by sketching them using the normal sketching tools and then adding the dimensions.

GENERATIVE SHAPE DESIGN

The **wire frame** and a **surface design** workbench is also a parametric and feature-based environment, in which you can create wireframe or surface models. The tools in this workbench are similar to those in the **Part Design** workbench. The only difference is that the tools in the environment are used to create basic and advanced surfaces. You are also provided with the surface editing tools, which are used to manipulate the surfaces to obtain the required shape.

SHEET METAL DESIGN

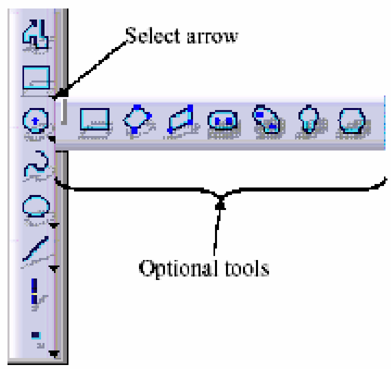
CATIA Sheet Metal Design is dedicated to the designing of sheet metal parts.

Sketcher Work Bench Tool Bars

There are three standard tool bars found in the **Sketcher Work Bench**. The three tool bars are shown below. The individual tools found in each of the three tool bars are labeled to the right of the tool icon.

Some tools have an arrow located at the bottom right of the tool icon. The arrow  is an indication that there is more than one variation of that particular type of tool. The tools that have more than one option are listed to the right of the default tool. To display the other tool options you must select and hold the left mouse button on the arrow as shown in Figure 1.1.

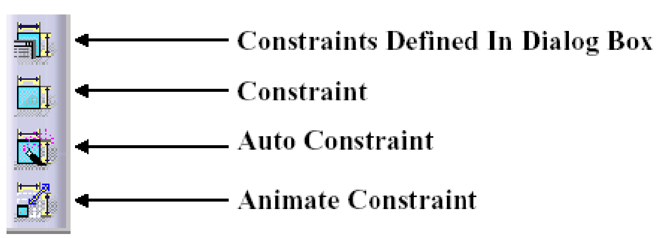
Figure 1.1



This will bring up the optional tools window. Move your mouse to the desired tool and release the mouse button. The desired tool icon now becomes the default tool, shown on the tool bar. All you have to do to select the new default tool is to double click on it.

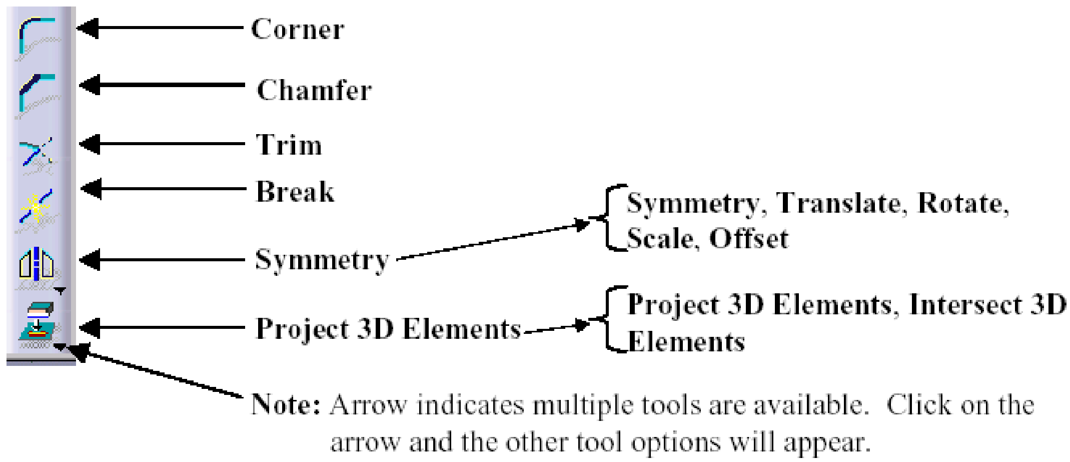
The **Constraints** Tool Bar

Tool Bar	Tool Name (default)	Tool Type Options



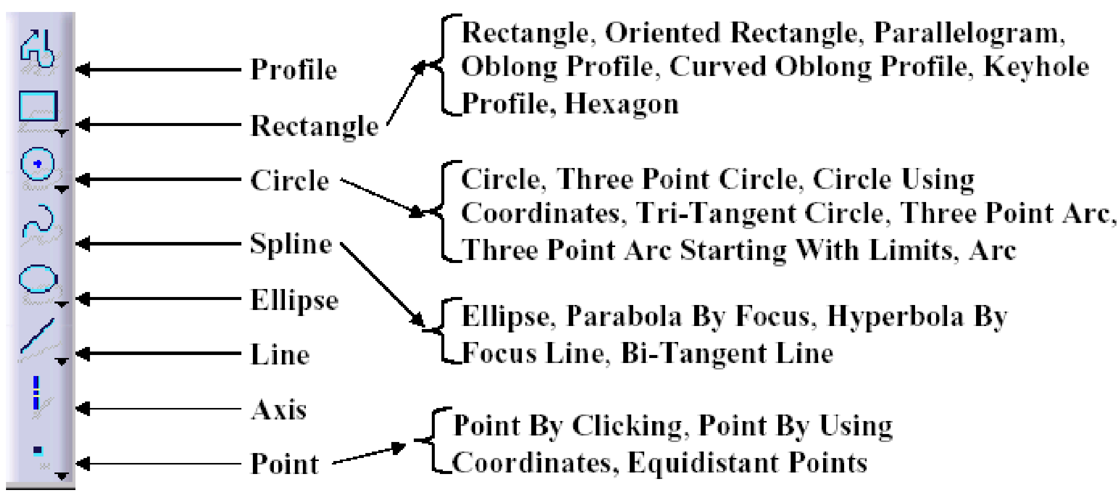
The **Operation** Tool Bar

Tool Bar **Tool Name** (default) **Tool Type Options**



The **Profile** Tool Bar

Tool Bar **Tool Name** (default) **Tool Type Options**



Experiment no: 1

DESIGN AND MODELLING OF RECTANGULAR PLATE WITH HOLE

Aim:

To draw the design and modeling of rectangular plate with hole using the CATIA package

Commands used:

- Lines
- Rectangle
- Circle
- Extrude or pad
- Pocketing
- Edge Fillet

Tools required

CATIA package

Procedure

1) Open the software tool CATIA package.
2) Create the new file and save it in the directory
3) Set the units and limits for the display screen
4) Use the line command and draw as per the given dimensions
5) Draw the circle by the given dimensions
6) Mark the dimensions for the riveted job
7) Save the diagram in specified directory

Result

Thus the rectangular plate with hole was designed and modeled as per the given dimension and verified

Experiment no: 2

DESIGN AND MODELLING OF SPAR COMPONENTS

Aim:

To draw the design and modeling of spar components using the CATIA package

Commands used:

- Lines
- Circle
- Extrude or pad
- Pocketing
- Multicopy
- Edge Fillet

Tools required

CATIA package

Procedure

1) Open the software tool CATIA package.
2) Create the new file and save it in the directory
3) Set the units and limits for the display screen
4) Use the line command and as per the given dimensions
5) Draw the spar assembly by the given dimensions
6) Mark the dimensions for the riveted job
7) Save the diagram in specified directory

Result

Thus the spar component was designed and modeled as per the given dimension and verified

Experiment no: 3

Design and Modeling of Aerofoil structures

Aim:

To draw the Design and Modeling of Aerofoil structures using the CATIA package

Commands used:

- Dots
- Spline
- Coordinate points
- Plane
- Loft

Tools required

CATIA package

Procedure:

1) Open the CATIA package.
2) Create the new file and save it in the directory
3) Set the units and limits for the display screen
4) Use the line command and draw as per the given dimensions
5) Import the aerofoil Coordinate points
6) Draw the drawing by the given dimensions
7) Mark the dimensions for the riveted job
8) Save the diagram in specified directory

Result

Thus the Aerofoil structures was designed and modeled as per the given dimension and verified

Experiment no: 4

Design and Modeling of cut section of wings

Aim:

To draw the Design and Modeling of cut section of wings using the CATIA package

Commands used:

- Spline
- Coordinate points
- Plane
- Loft

Tools required

CATIA package

Procedure

1) Open the CATIA package.
2) Create the new file and save it in the directory
3) Set the units and limits for the display screen
4) Use the line command and draw the as per the given dimensions
5) Draw the diagram by the given dimensions
6) Mark the dimensions for the riveted job
7) Save the diagram in specified directory

Result

Thus the required drawing was designed and modeled as per the given dimension and verified

Experiment no: 5

Design and Modeling of machine components

Aim:

To draw the Design and Modeling of machine components using the CATIA package

Commands used:

- Lines
- Groove
- Shaft
- Plane
- Pocketing
- Translate

Tools required

CATIA package

Procedure

1) Open the CATIA package.
2) Create the new file and save it in the directory
3) Set the units and limits for the display screen
4) Use the line command and draw the as per the given dimensions
5) Draw the diagram using commands by the given dimensions
6) Mark the dimensions Save the diagram in specified directory

Result

Thus the required drawing was designed and modeled as per the given dimension and verified

Experiment no: 6

Design and Modeling of bulkhead

Aim:

To draw the Design and Modeling of bulk head using the CATIA package

Commands used:

- Lines
- Groove
- Shaft
- Plane
- Pocketing
- Translate

Tools required

CATIA package

Procedure

1) Open the software tool CATIA package.
2) Create the new file and save it in the directory
3) Set the units and limits for the display screen
4) Use the line command and draw the as per the given dimensions
5) Draw the bulk head assembly by the given dimensions
6) Mark the dimensions for the riveted job
7) Save the diagram in specified directory

Result

Thus the bulk head assembly was designed and modeled as per the given dimension and verified

ANALYSIS PART

INTRODUCTION

Structural analysis is the determination of the effects of loads on physical structures and their components. Structures subject to this type of analysis include all that must withstand loads, such as buildings, bridges, vehicles, machinery, furniture, attire, soil strata, prostheses and biological tissue. Structural analysis incorporates the fields of applied mechanics, materials science and applied mathematics to compute a structure's deformations, internal forces, stresses, support reactions, accelerations, and stability. The results of the analysis are used to verify a structure's fitness for use, often saving physical tests. Structural analysis is thus a key part of the engineering design of structures.

1.2 Classification of Structures

All structural forms used for load transfer from one point to another are 3-dimensional in nature. In principle one could model them as 3-dimensional elastic structure and obtain solutions (response of structures to loads) by solving the associated partial differential equations. In due course of time, you will appreciate the difficulty associated with the 3-dimensional analysis. Also, in many of the structures, one or two dimensions are smaller than other dimensions. This geometrical feature can be exploited from the analysis point of view. The dimensional reduction will greatly reduce the complexity of associated governing equations from 3 to 2 or even to one dimension. This is indeed at a cost. This reduction is achieved by making certain assumptions (like Bernoulli-Euler' kinematic assumption in the case of beam theory) based on its observed behavior under loads. Structures may be classified as 3-, 2- and 1-dimensional (see Fig. 1.1(a) and (b)). This simplification will yield results of reasonable and acceptable accuracy. Most commonly used structural forms for load transfer are: beams, plane truss, space truss, plane frame, space frame, arches, cables, plates and shells. Each one of these structural arrangement supports load in a specific way.

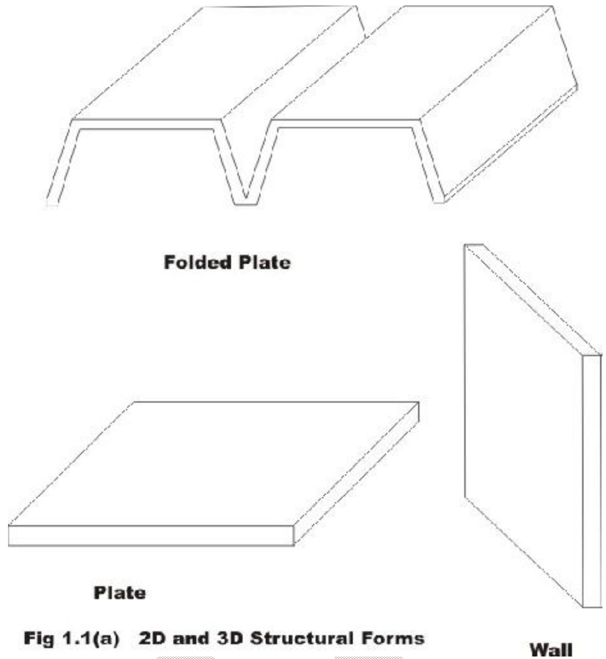


Fig 1.1(a) 2D and 3D Structural Forms

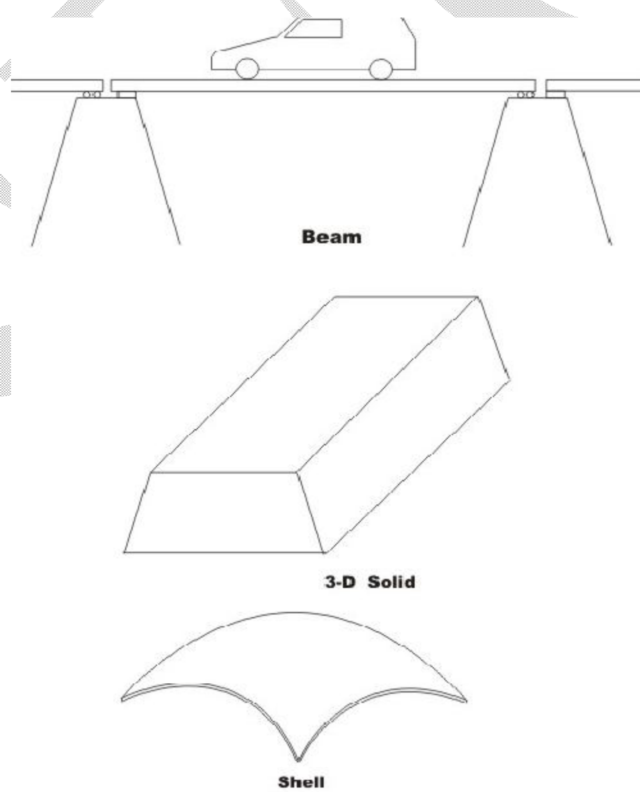


Fig 1.1(b) Commonly Used Structural Forms

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Beams are the simplest structural elements that are used extensively to support loads. They may be straight or curved ones. For example, the one shown in Fig. 1.2 (a) is hinged at the left support and is supported on roller at the right end. Usually, the loads are assumed to act on the beam in a plane containing the axis of symmetry of the cross section and the beam axis. The beams may be supported on two or more supports as shown in Fig. 1.2(b). The beams may be curved in plan as shown in Fig. 1.2(c). Beams carry loads by deflecting in the same plane and it does not twist. It is possible for the beam to have no axis of symmetry. In such cases, one needs to consider unsymmetrical bending of beams. In general, the internal stresses at any cross section of the beam are: bending moment, shear force and axial force.

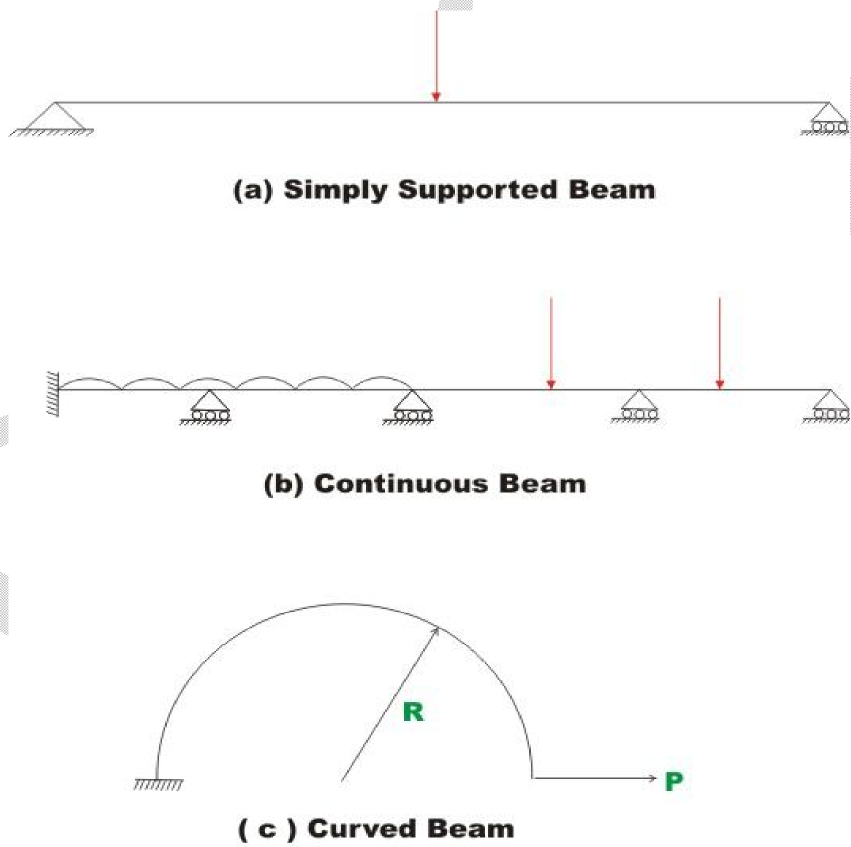
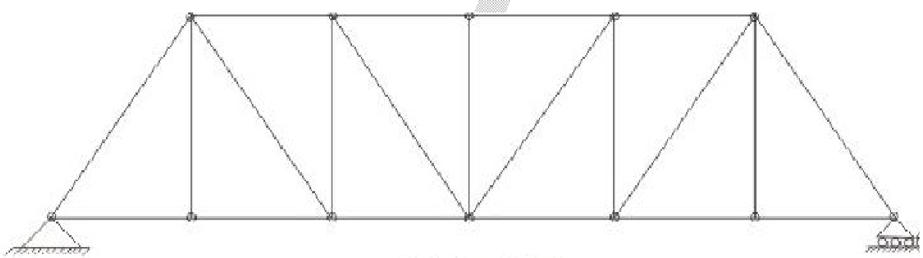
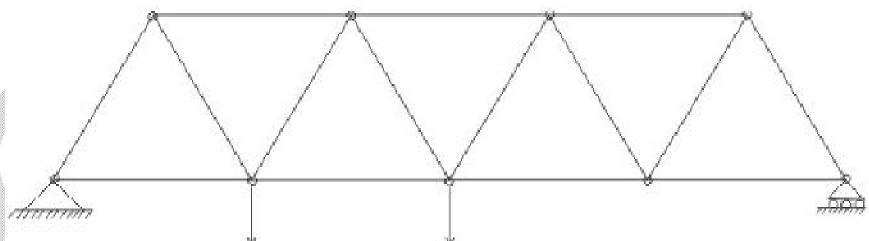


Fig 1.2 Beams

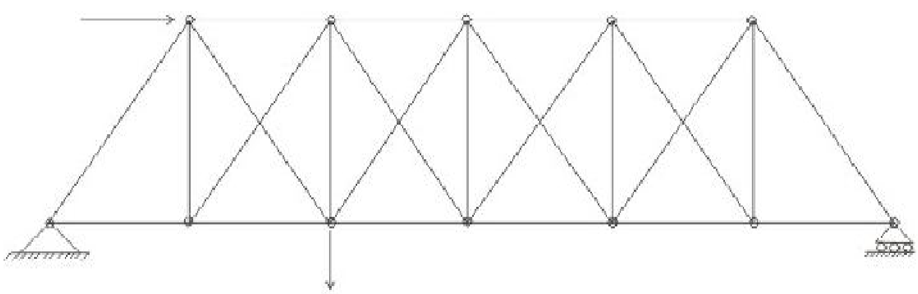
In India, one could see **plane trusses** Fig. 1.3 (a),(b),(c) commonly in Railway bridges, at railway stations, and factories. Plane trusses are made of short thin members interconnected at hinges into triangulated patterns. For the purpose of analysis statically equivalent loads are applied at joints. From the above definition of truss, it is clear that the members are subjected to only axial forces and they are constant along their length. Also, the truss can have only hinged and roller supports. In field, usually joints are constructed as rigid by welding. However, analyses were carried out as though they were pinned. This is justified as the bending moments introduced due to joint rigidity in trusses are negligible. Truss joint could move either horizontally or vertically or combination of them. In ***space truss*** (Fig. 1.3 (d)), members may be oriented in any direction. However, members are subjected to only tensile or compressive stresses. Crane is an example of space truss.



(a) Pratt Truss



(b) Warren Truss



(c) Double Warren Truss

Fig 1.3 Trusses

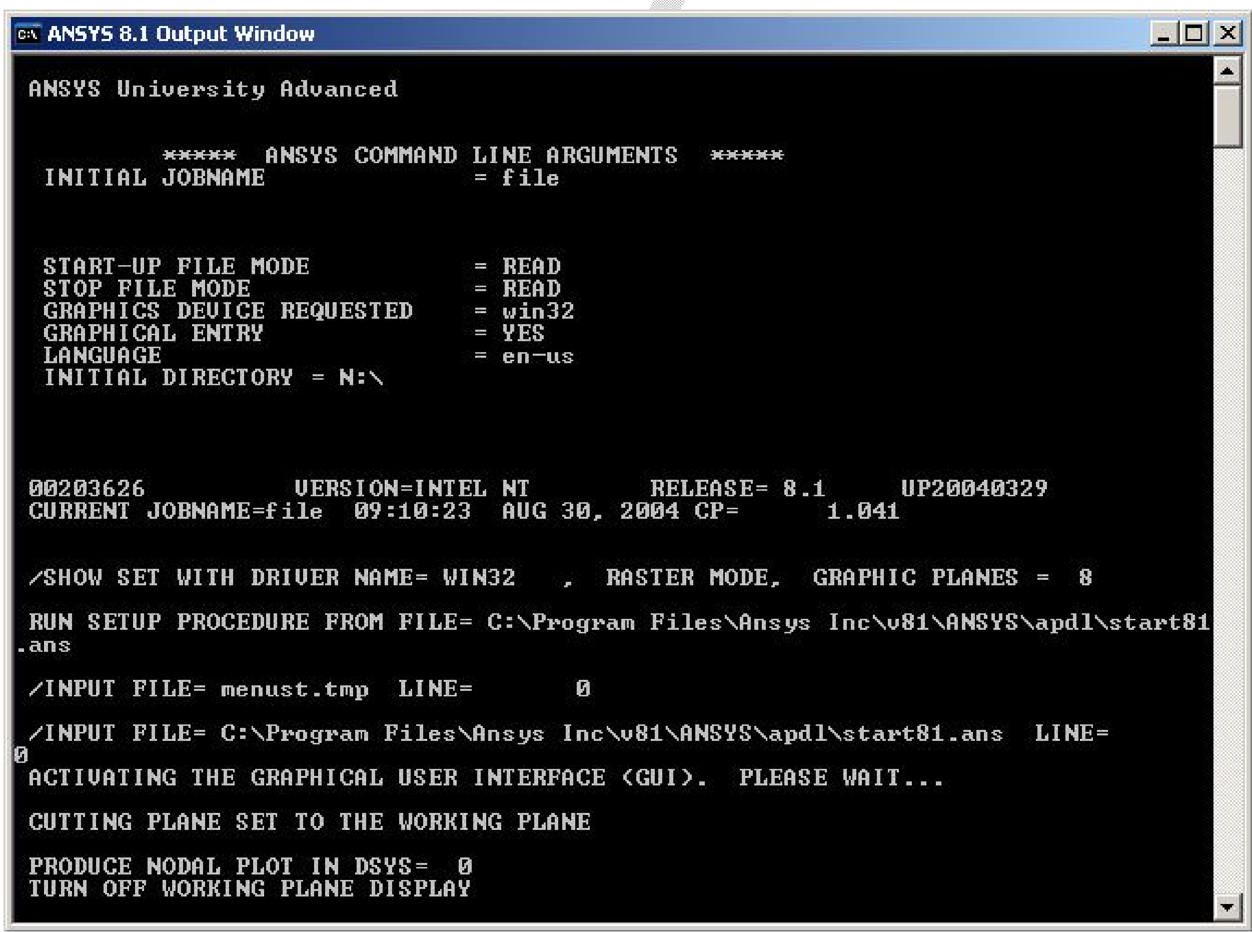
Introduction to ANSYS

Starting ANSYS:

The ANSYS graphical user interface can be started by selecting the ANSYS icon located in the ANSYS 8.1 folder. Selecting the ANSYS icon will take you directly to the graphical user interface.

ANSYS Graphical User Interface:

After starting ANSYS, two windows will appear. The first is the ANSYS 8.1 Output Window:

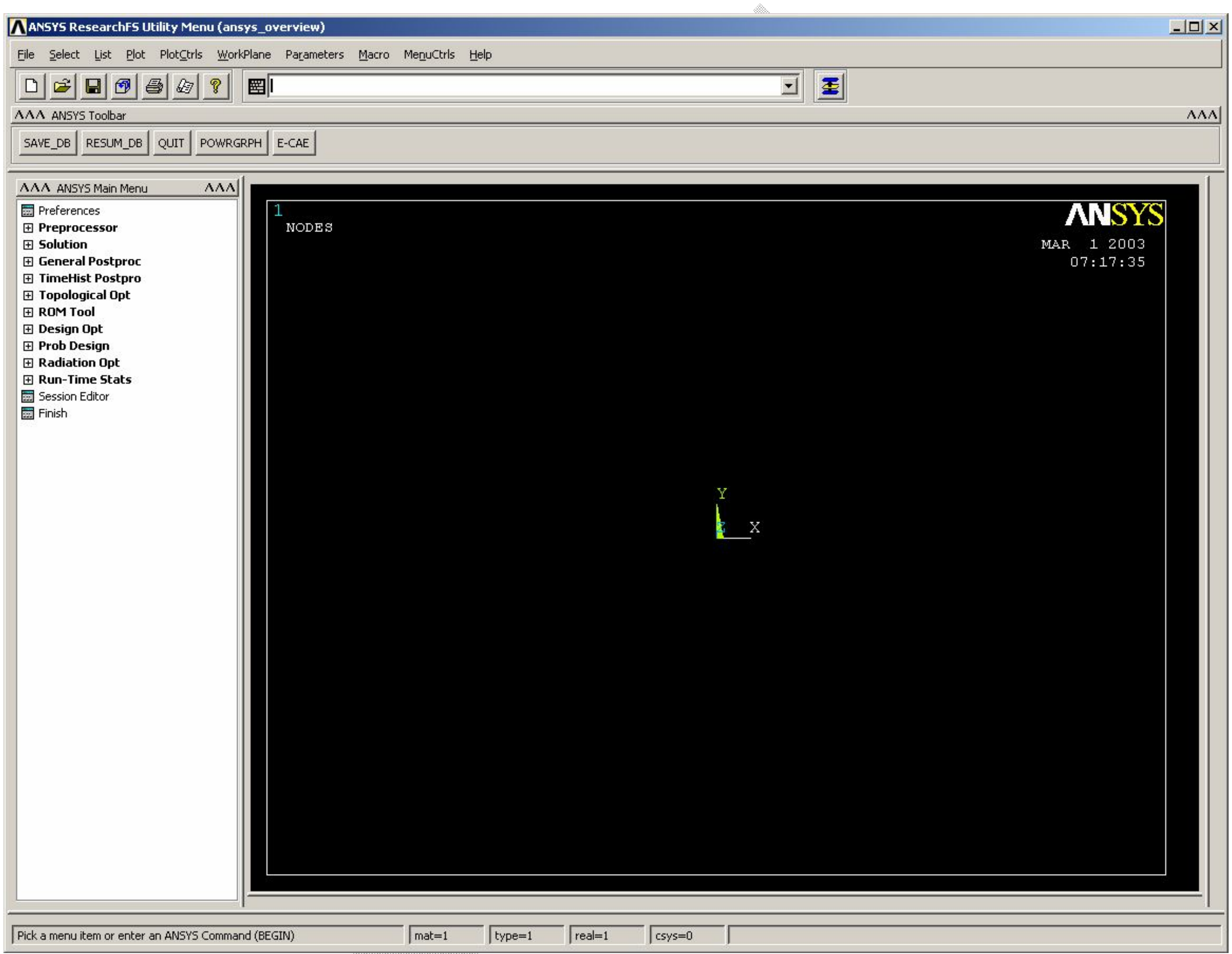


This window displays a listing of every command that ANSYS executes. If you encounter problems, this is a good place to look to see what ANSYS is doing or has done. This is one location where you will find all of the warnings and error messages that appear and the command that generated the warning/error.

The second window is the ANSYS Research FS graphical user interface. This is divided into 4 sections

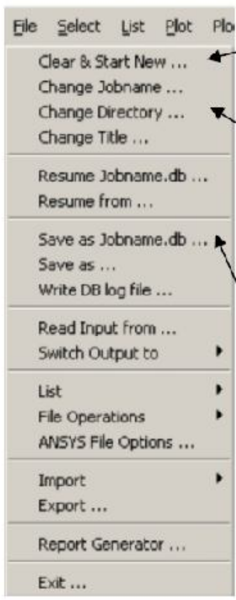
1. ANSYS Utility Menu
2. ANSYS Toolbar Menu
3. ANSYS Main Menu
4. Display window

Each section will be discussed in further detail below.



File Drop-down Menu

The File drop-down menu includes the options to clear the database, change, resume, and save the current model.



Clear and Start New deletes the current database. It does not clear the log or error files.

Change Jobname changes the name of the database and associated files. The next time you save, it will write everything to the current jobname. It will not delete the previous jobname or associated files.
Note: unless you check the box for New log and error files, it will continue to write to the current log and error files.

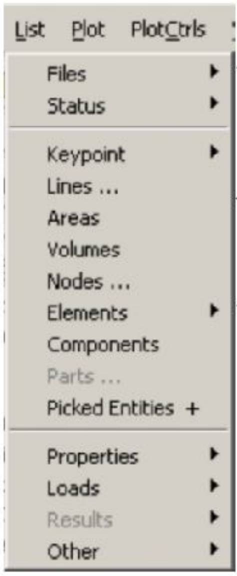
Change Directory allows you to switch directories where the files are being saved.

Resume Jobname.db and Resume from allows you to open a model that has already been saved. Note: If you resume a file, ANSYS does not automatically switch the current jobname to the name of the file you resumed from. Change the jobname otherwise you may write over another model.

Save as Jobname.db and Save as allow you to save the model. If you choose "save as" it will save the file as a different name. If you do not change the jobname, the next time you save it will overwrite the current jobname and not the "save as" file name.

List Pull-down Menu

The list pull-down menu allows you to view the log and error files, obtain a listing of geometric entities, elements and their properties, nodes, and boundary conditions and loads applied to the model.



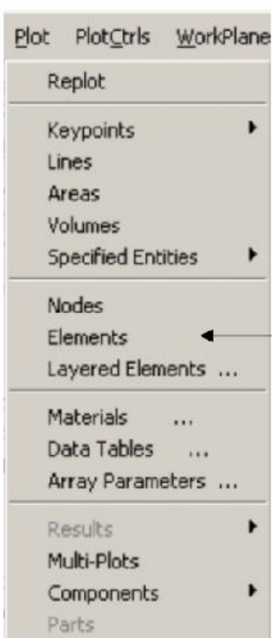
This menu allows you to view the log and error files. The log file is a summary of every command performed in ANSYS while the error file may provide insight into troubleshooting problems with the model/analysis.

These list the geometric entities and their components.

Selecting the Nodes and Elements listings are easy methods to determine the number of nodes and elements in the model.

Plot Pull-down Menu

This pull-down menu allows you to plot the various components of the model such as keypoints, areas, volumes and elements.



Use the plot elements command to obtain the mesh plot.

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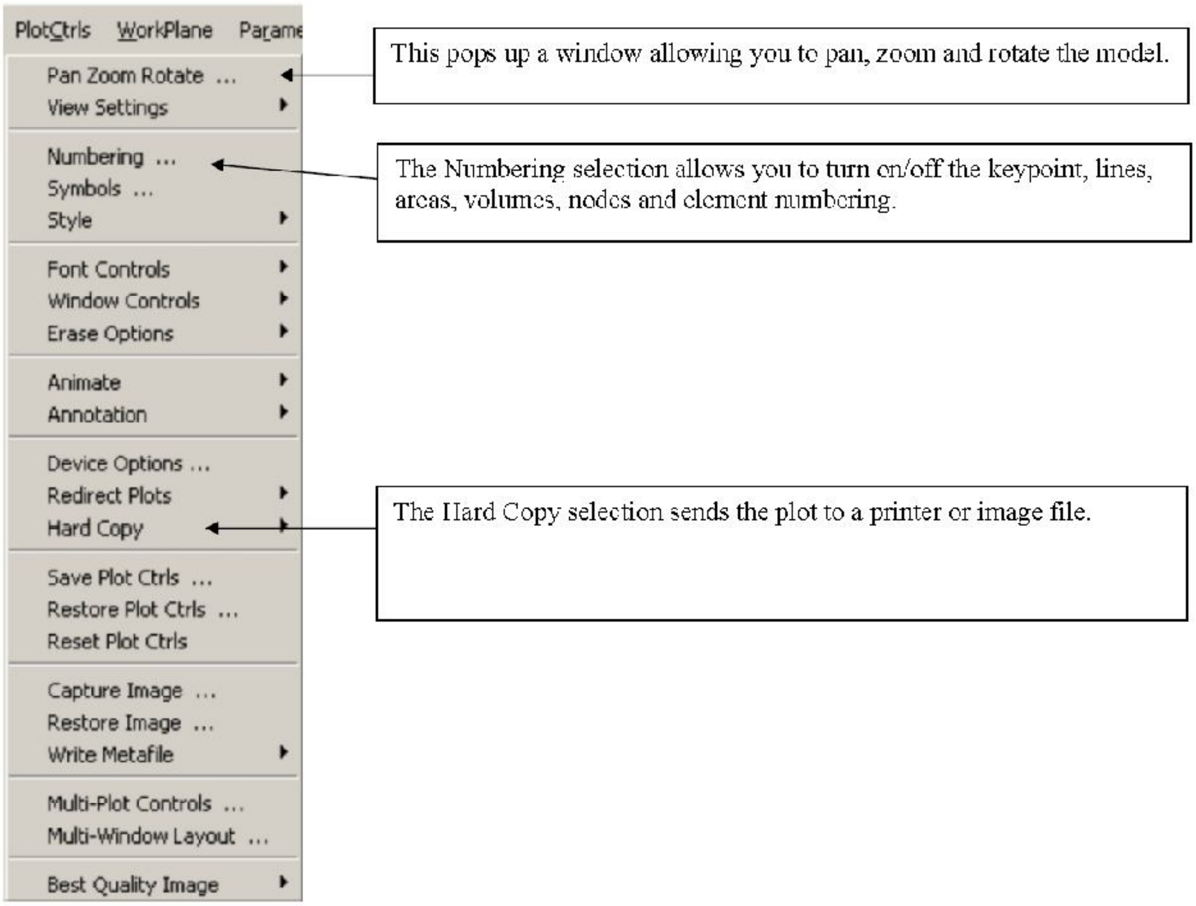
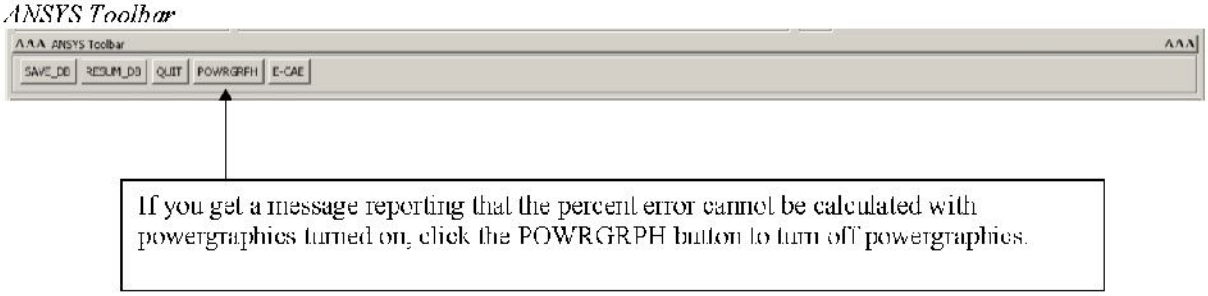
PlotCtrls Pull-Down Menu

This menu includes the controls to pan/zoom/rotate your model, select the numbering options, change styles and generate hard copies of the plots.



ANSYS Toolbar



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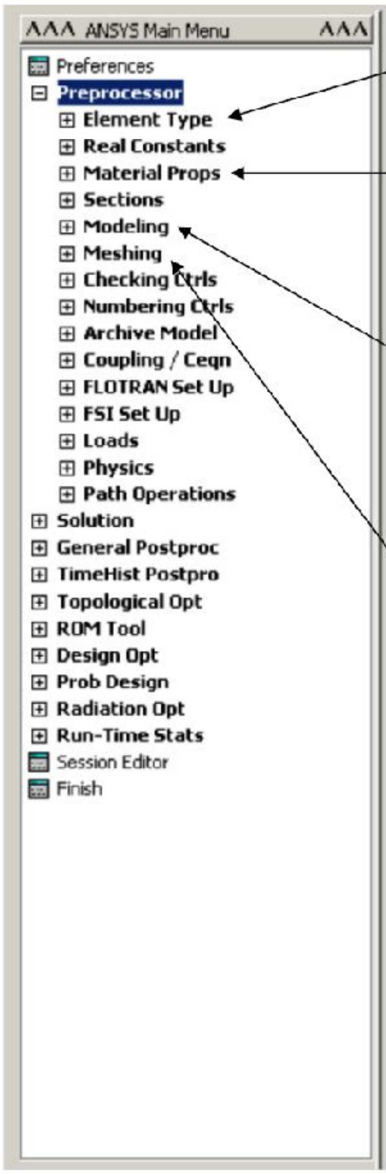
ANSYS Main Menu

The ANSYS Main Menu contains all of the commands to create, mesh, apply loads, solve, and view results of the FE analysis. The Main Menu is divided into sections that sequentially follow the steps involved in an analysis.

Preprocessor



Select the element types to be used. Multiple types may be selected and applied to a model.

Input the material properties. Multiple materials may be entered and applied to a model.

Use the modeling options to create the model. Note: ANSYS uses a hierarchy for modeling:

keypoints (lowest)
lines
areas
volumes (highest)

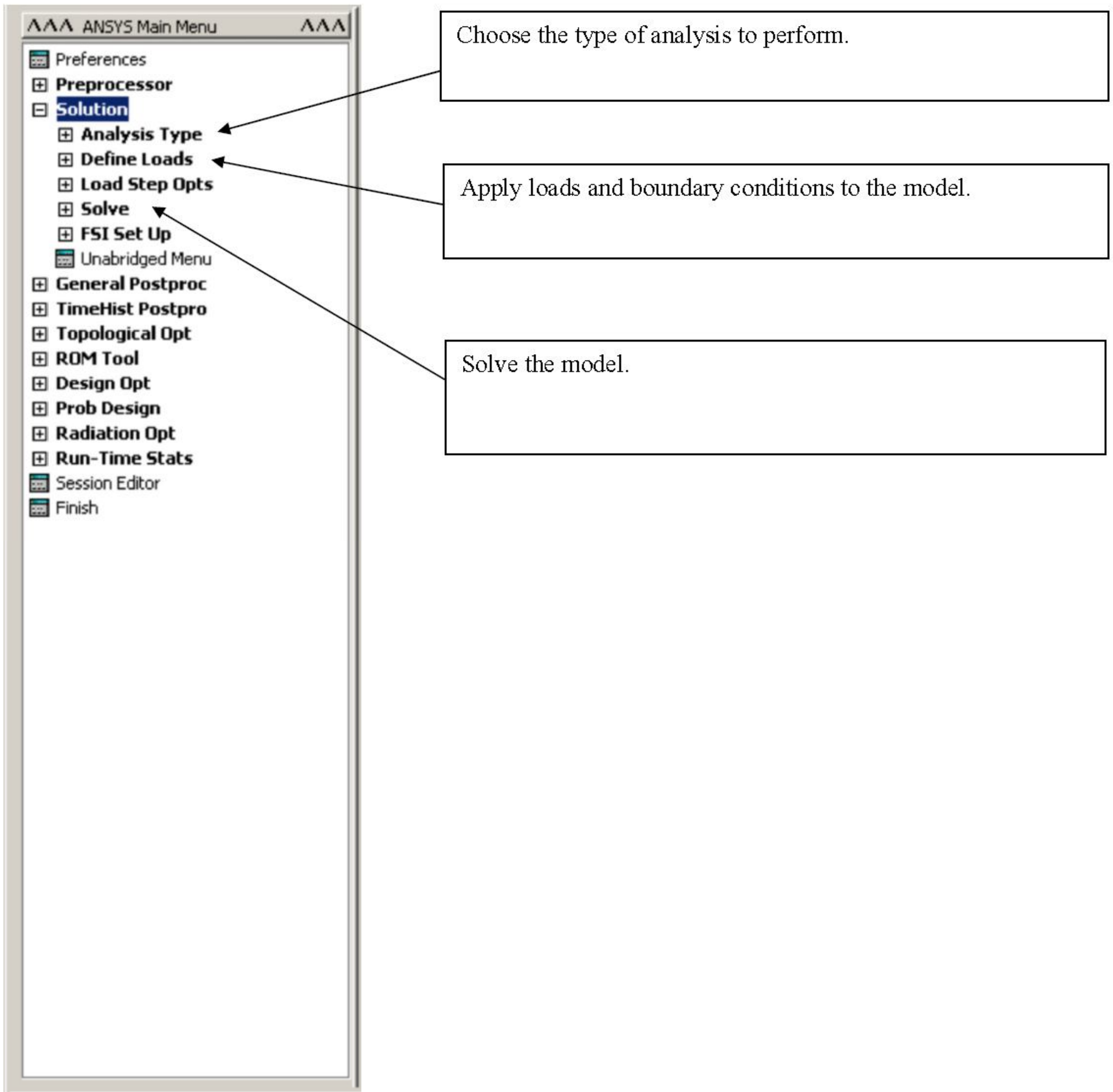
If you create a volume, all of the areas, lines and keypoints will be automatically created for you.

Use the meshing options to mesh the model.

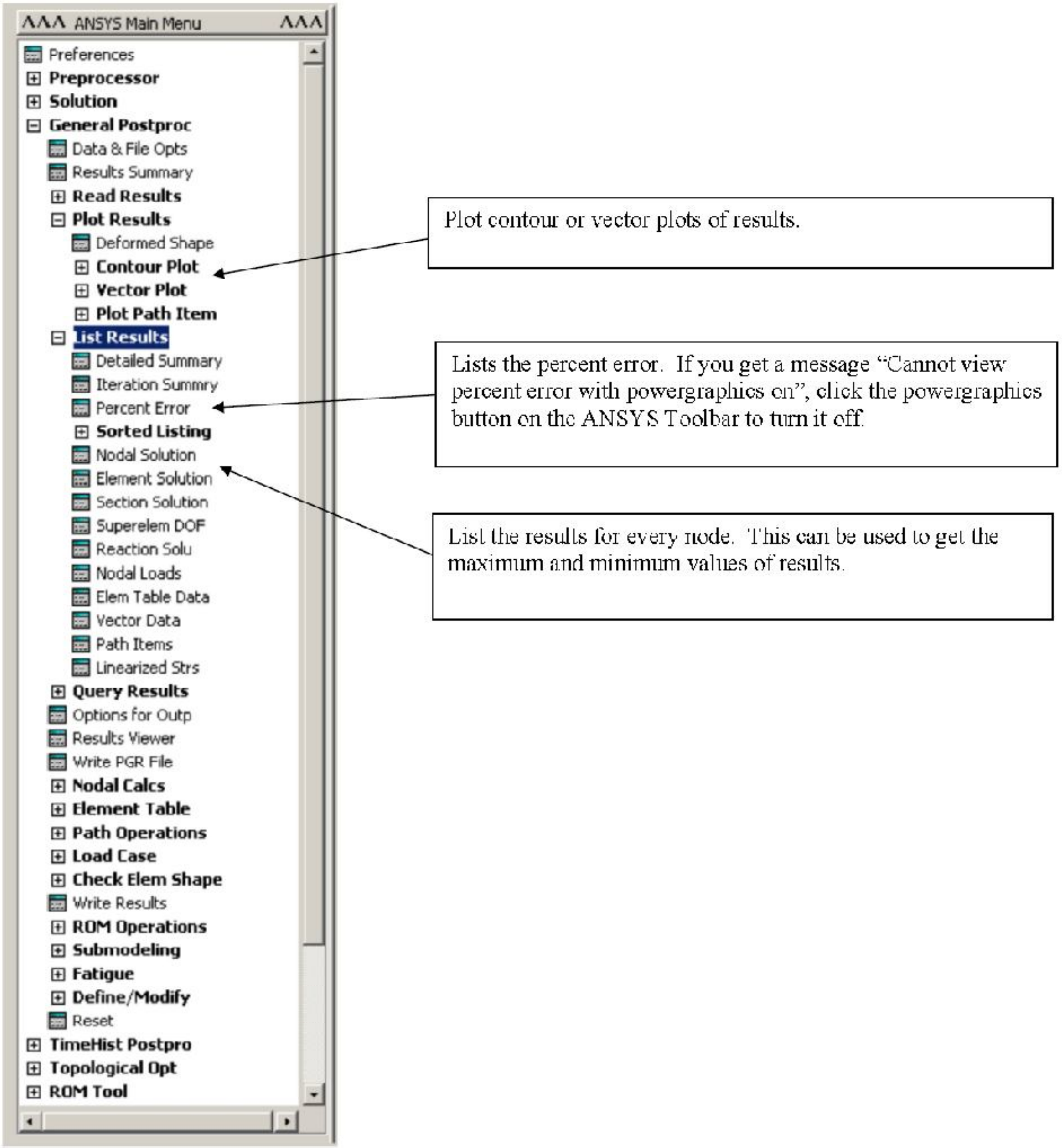
To delete mistakes, choose the Delete menu under Modeling. It is important to select the right option. For example, if you select Areas Only, the area will be deleted but the lines and keypoints that make up that area will remain. If you select Areas and Below, it will delete the area plus the lines and keypoints that define the area.

NOTE: there is not an undo button in ANSYS, once deleted, the component must be re-created.

Solution


ANSYS Main Menu
Preferences
Preprocessor
Solution
Analysis Type
Define Loads
Load Step Opts
Solve
FSI Set Up
Unabridged Menu
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
Design Opt
Prob Design
Radiation Opt
Run-Time Stats
Session Editor
Finish
Choose the type of analysis to perform.
Apply loads and boundary conditions to the model.
Solve the model.

Post Processing

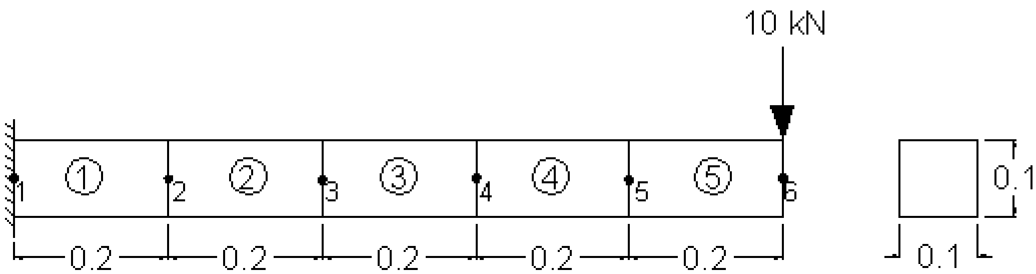


Experiment no: 7

Design and analysis of cantilever beam.

AIM:

To analyze the deflection of a cantilever beam with point load at its free end using ANSYS.



(All dimensions are in 'm')

Material Properties:

Young's modulus, $E=2x10^8 kN/m^2$

Poisson ratio, $\mu=0.3$

Moment of inertia, $I_{zz} = (bd^3/12) = 8.33x10^{-6} m^4$

Element Type:

2D Elastic Beam

PROCEDURE:

1. Specify the type of analysis

Preferences > structural > ok

2. Specify the element type

Pre-processor > element type > Add/Edit/Delete > Add > beam > 2D elastic 3>ok

3. Specify real constants

Pre-processor > real constants > Add/Edit/Delete > Add (specify cross sectional area, moment of inertia, beam height) > ok

4. **Specify material properties**

Pre-processer > metal properties > material model > structural > linear > elastic > isotropic (Specify young's modules & Poisson ratio) > ok

5. Create node and element

Pre-processor > modelling> create > nodes > Inactive CS >specify the nodes in the Element>OK

Pre-processer > modelling > create > elements > Auto numbered > thru nodes >

Pick nodes (1&2, 2&3, 3&4, 4&5, 5&6) > ok

6. Apply Constraints and Forces

Pre-processor> loads > define load > apply > structural > displacement > on node > select node1 > All DOF> OK

Pre-processor > loads >define loads >apply >structural >force/moments >on nodes > select node 6 >specify force value(Fy) >OK.

7. Solving the model

Solution >solve >current LS

VIEWING RESULTS:

Nodal displacements & stresses at node

General post processor >element table >define table >Add >by sequence number > NMSIC >1 >apply

[specify for 3&2,4 also]

General post processor >element table >list element table >select name >OK
The result is displayed as shown in table 2.

General post processor >list results >nodal solution >DOF solution >Uy >OK
The result is displayed as shown in table 1.

REPORT:

The deflection of a cantilever beam with point load at free end was analysed using ANSYS.

Experiment no: 8

DESIGN AND ANALYSIS OF A TRUSS 1.

AIM

To analyse the deflection of a given simple stress subjected to given load and constraints using ANSYS.

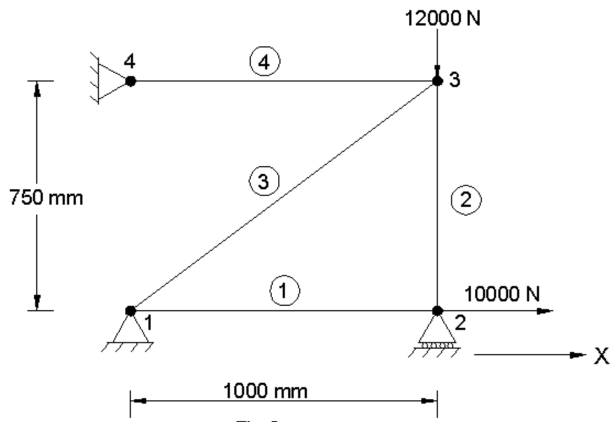


Fig.2

It is given that $E = 2 \times 10^5$ N/mm^2 and $A_e = 625$ mm^2 for all elements

PROCEDURE:

PREPROCESSING: DEFINING THE PROBLEM

1. **Give example a Title**
   Preference > structural >ok.

2. **Define Element Types**
   Preprocessor >element type > add/edit/del > add >link > link 180 > ok > close.

3. **Define Real Constants**
   Preprocessor > add/edit/del>add>ok > put area 1 > ok > close.

4. **Define Material Property**
   Preprocessor > material properties > material model (1) > structural> linear > elastic > isotropic > put (E) value > ok > close.

MODELING AND MATERIAL ASSIGNING

1. **Create modeling**
   Preprocessor > modeling > create > nodes > in active CS > x=0,Y=0 > apply > x=1000,Y=0 >apply > x=1000,Y=750 > apply > x=0,Y=750 > ok

Preprocessor > Modeling > create > elements > auto numbered > thru nodes > click node 1&2 > apply > click node 2&3 > click node 1&3 > apply > click node 3&4 >ok

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

APPLY LOADS AND CONSTRAINTS:

Preprocessor > loads > define loads > apply > structural > displacement > on nodes > pick node1 > ok > select all DOF > ok

Preprocessor > loads > define loads > apply > structural > displacement > on nodes > pick node2 > ok > select Uy > ok

Preprocessor > loads > define load > apply > structural > force/moment > on nodes > select node2 > ok > select direction Fx > put force value(10000) > ok

Preprocessor > loads > define load > apply > structural > force/moment > on nodes > select node3 > ok > select direction Fy > put force value(-12000) > ok

Solution> solve > current LS > ok > close.

POSTPROCESSING: VIEWING THE RESULTS

To Find The Numerical Value:

General postprocessor > list result > nodal solution > select DOF solution > select displacement vector sum > ok.

To Find The Graphical Value:

General postprocessor > Plot results > contour plot > nodal solution > select DOF solution > select displacement vector sum > ok.

REPORT:

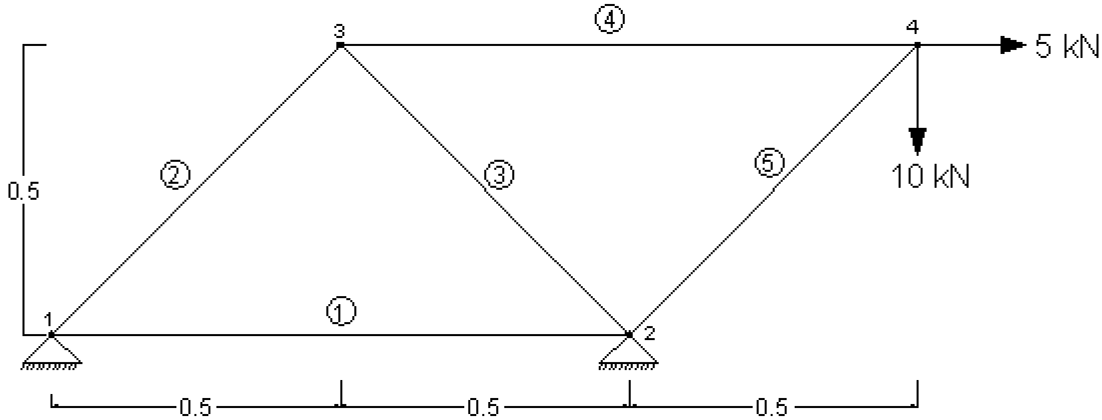
Thus the given experiment was done and results were verified.

Experiment no: 9

DESIGN AND ANALYSIS OF A TRUSS 2.

AIM

To analyse the deflection of a given simple stress subjected to given load and constraints using ANSYS.



All dimensions are in 'm'

It is given that $E = 2 \times 10^8$ N/m^2 , $\mu = 0.3$ and $A_e = 0.1$ m^2 for all elements

PROCEDURE:

PREPROCESSING: DEFINING THE PROBLEM

1. **Give example a Title**
   Preference > structural >ok.

2. **Define Element Types**
   Preprocessor >element type > add/edit/del > add >link > link 180 > ok > close.

3. **Define Real Constants**
   Preprocessor > add/edit/del>add>ok > put area 1 > ok > close.

4. **Define Material Property**
   Preprocessor > material properties > material model (1) > structural> linear > elastic > isotropic > put (E) value >put Poisson ratio(μ) >ok > close.

MODELING AND MATERIAL ASSIGNING

1. **Create modeling**

Preprocessor > modeling > create > nodes > in active CS > x=0,Y=0 > apply > x=1,Y=0 >apply > x=0.5,Y=0.5 > apply > x=1.5,Y=0.5 > ok

Preprocessor > Modeling > create > elements > auto numbered > thru nodes > click node 1&2 > apply > click node 2&3 > click node 1&3 > apply > click node 3&4 >ok

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

APPLY LOADS AND CONSTRAINTS:

Preprocessor > loads > define loads > apply > structural > displacement > on nodes > pick node 1&2 > ok > select all DOF > ok

Preprocessor > loads > define load > apply > structural > force/moment > on nodes > select
Node4 > ok > select direction Fx > put force value(5000) > ok

Preprocessor > loads > define load > apply > structural > force/moment > on nodes > select
Node4 > ok > select direction Fy > put force value(-10000) > ok

Solution> solve > current LS > ok > close.

POSTPROCESSING: VIEWING THE RESULTS

To Find The Numerical Value:

General postprocessor > list result > nodal solution > select DOF solution > select displacement vector sum > ok.

To Find The Graphical Value:

General postprocessor > Plot results > contour plot > nodal solution > select DOF solution > select displacement vector sum > ok.

REPORT:

Thus the given experiment was done and results were verified.

CAM PART

COMPUTER-AIDED MANUFACTURING

Computer-aided manufacturing (**CAM**) is the use of computer-based software tools that assist engineers and machinists in manufacturing or prototyping product components. Its primary purpose is to create a faster production process and components with more precise dimensions and material consistency, which in some cases, uses only the required amount of raw material (thus minimizing waste), while simultaneously reducing energy consumption. CAM is a programming tool that makes it possible to manufacture physical models using computer-aided design (CAD) programs. CAM creates real life versions of components designed within a software package.

CNC Technology

Numerical Control (NC) is a software-based machine tool control technique developed at Massachusetts Institute of Technology (MIT) in early 1960s. It has now evolved into a mature technology known as Computer Numerical Control (CNC). Although major applications of CNC even today continue to be in machining, it finds applications in other processes such as sheet metal working, non-traditional machining and inspection. Robots and Rapid Prototyping machines are also CNC controlled. In fact, any process that can be visualized as a sequence of motions and switching functions can be controlled by CNC. These motions and switching functions are input in the form of alphanumeric instructions. CNC is the basis of flexible automation which helps industries cut down time-to-market and enables launch of even low volume products. Unlimited muscle power, unmanned operation, independent axes coordinated through software, simplified generic tooling even for the most complex jobs and accurate construction are some of the salient features of CNC.

CNC Machining

Automats and Special Purpose Machines (SPMs) require special cams/ templates and clutch settings for each part. Manufacture of these cams/ templates is costly and slow.

Furthermore, changing over from one part to the other on these machines also consumes considerable time. The high cost and long time of these hard automated machines to produce parts can be justified only in mass production. With the advent of fast, rigid and accurate CNC machines and sophisticated CAM packages, generation of NC programs and change over from one product to the other are easy and fast as it does not require any mechanical change. These in conjunction with advanced cutting tools have made High Speed Cutting (HSC) of hard materials a reality. Therefore, CNC machining has become a standard means to produce dies and molds; tool makers today require EDM only for producing inaccessible regions, sharp corners, tiny features and desired surface quality. Intricate aerospace parts are realized through 5 axis CNC machining. Internet technology in a global village enables designing in one place, NC programming and verification in another place and actual machining in yet another place.

Advantages of CNC

- Flexibility
- Accuracy
- Speed
- Simplified fixturing and generic cutting tools
- Storage of machining skill in NC programs
- Less skilled operators will do
- Less fatigue to the operators

G-codes

G-Code, or preparatory code or function, are functions in the Numerical control programming language. The G-codes are the codes that position the tool and do the actual work, as opposed to M-codes, that manages the machine; T for tool-related codes. S and F are tool-Speed and tool-Feed, and finally D-codes for tool compensation. The programming language of Numerical Control (NC) is sometimes informally called **G-code**. But in actuality, G-codes are only a part of the NC-programming language that controls NC and CNC machine tools. A basic list of `G' operation codes is given below.

1. G00 - Rapid move (not cutting)
2. G01 - Linear move
3. G02 - Clockwise circular motion
4. G03 - Counterclockwise circular motion

5. G04 - Dwell
6. G05 - Pause (for operator intervention)
7. G08 - Acceleration
8. G09 - Deceleration
9. G17 - x-y plane for circular interpolation
10. G18 - z-x plane for circular interpolation
11. G19 - y-z plane for circular interpolation
12. G20 - turning cycle or inch data specification
13. G21 - thread cutting cycle or metric data specification
14. G24 - face turning cycle
15. G25 - wait for input #1 to go low (Prolight Mill)
16. G26 - wait for input #1 to go high (Prolight Mill)
17. G28 - return to reference point
18. G29 - return from reference point
19. G31 - Stop on input (INROB1 is high) (Prolight Mill)
20. G33-35 - thread cutting functions (Emco Lathe)
21. G35 - wait for input #2 to go low (Prolight Mill)
22. G36 - wait for input #2 to go high (Prolight Mill)
23. G40 - cutter compensation cancel
24. G41 - cutter compensation to the left
25. G42 - cutter compensation to the right
26. G43 - tool length compensation, positive
27. G44 - tool length compensation, negative
28. G50 - Preset position
29. G70 - set inch based units or finishing cycle
30. G71 - set metric units or stock removal
31. G72 - indicate finishing cycle (EMCO Lathe)
32. G72 - 3D circular interpolation clockwise (Prolight Mill)
33. G73 - turning cycle contour (EMCO Lathe)
34. G73 - 3D circular interpolation counter clockwise (Prolight Mill)
35. G74 - facing cycle contour (Emco Lathe)
36. G74.1 - disable 360 deg arcs (Prolight Mill)
37. G75 - pattern repeating (Emco Lathe)
38. G75.1 - enable 360 degree arcs (Prolight Mill)
39. G76 - deep hole drilling, cut cycle in z-axis
40. G77 - cut-in cycle in x-axis
41. G78 - multiple threading cycle

42. G80 - fixed cycle cancel
43. G81-89 - fixed cycles specified by machine tool manufacturers
44. G81 - drilling cycle (Prolight Mill)
45. G82 - straight drilling cycle with dwell (Prolight Mill)
46. G83 - drilling cycle (EMCO Lathe)
47. G83 - peck drilling cycle (Prolight Mill)
48. G84 - taping cycle (EMCO Lathe)
49. G85 - reaming cycle (EMCO Lathe)
50. G85 - boring cycle (Prolight mill)
51. G86 - boring with spindle off and dwell cycle (Prolight Mill)
52. G89 - boring cycle with dwell (Prolight Mill)
53. G90 - absolute dimension program
54. G91 - incremental dimensions
55. G92 - Spindle speed limit
56. G93 - Coordinate system setting
57. G94 - Feed rate in ipm (EMCO Lathe)
58. G95 - Feed rate in ipr (EMCO Lathe)
59. G96 - Surface cutting speed (EMCO Lathe)
60. G97 - Rotational speed rpm (EMCO Lathe)
61. G98 - withdraw the tool to the starting point or feed per minute
62. G99 - withdraw the tool to a safe plane or feed per revolution
63. G101 - Spline interpolation (Prolight Mill)

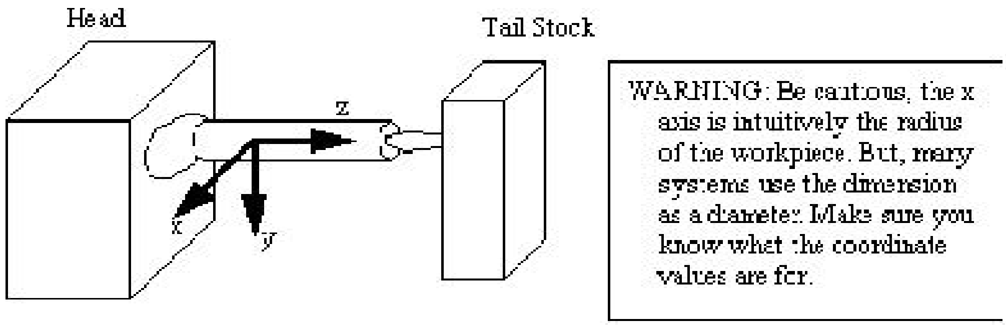
M-Codes

1. M-Codes control machine functions and these include,
2. M00 - program stop
3. M01 - optional stop using stop button
4. M02 - end of program
5. M03 - spindle on CW
6. M04 - spindle on CCW
7. M05 - spindle off
8. M06 - tool change
9. M07 - flood with coolant
10. M08 - mist with coolant
11. M08 - turn on accessory #1 (120VAC outlet) (Prolight Mill)
12. M09 - coolant off
13. M09 - turn off accessory #1 (120VAC outlet) (Prolight Mill)
14. M10 - turn on accessory #2 (120VAC outlet) (Prolight Mill)

15. M11 - turn off accessory #2 (120VAC outlet) (Prolight Mill) or tool change
16. M17 - subroutine end
17. M20 - tailstock back (EMCO Lathe)
18. M20 - Chain to next program (Prolight Mill)
19. M21 - tailstock forward (EMCO Lathe)
20. M22 - Write current position to data file (Prolight Mill)
21. M25 - open chuck (EMCO Lathe)
22. M25 - set output #1 off (Prolight Mill)
23. M26 - close chuck (EMCO Lathe)
24. M26 - set output #1 on (Prolight Mill)
25. M30 - end of tape (rewind)
26. M35 - set output #2 off (Prolight Mill)
27. M36 - set output #2 on (Prolight Mill)
28. M38 - put stepper motors on low power standby (Prolight Mill)
29. M47 - restart a program continuously, or a fixed number of times (Prolight Mill)
30. M71 - puff blowing on (EMCO Lathe)
31. M72 - puff blowing off (EMCO Lathe)
32. M96 - compensate for rounded external curves
33. M97 - compensate for sharp external curves
34. M98 - subprogram call
35. M99 - return from subprogram, jump instruction
36. M101 - move x-axis home (Prolight Mill)
37. M102 - move y-axis home (Prolight Mill)
38. M103 - move z-axis home (Prolight Mill)

CNC PROGRAMMING

The coordinates are almost exclusively Cartesian and the origin is on the work piece. For a lathe, the in feed/radial axis is the x-axis, the carriage/length axis is the z-axis. There is no need for a y-axis because the tool moves in a plane through the rotational center of the work. Coordinates on the work piece shown below are relative to the work.



CNC lathe / CNC turning center

CNC lathes are rapidly replacing the older production lathes (multispindle, etc) due to their ease of setting and operation. They are designed to use modern carbide tooling and fully utilize modern processes. The part may be designed and the tool paths programmed by the CAD/CAM process, and the resulting file uploaded to the machine, and once set and trailed the machine will continue to turn out parts under the occasional supervision of an operator. The machine is controlled electronically via a computer menu style interface; the program may be modified and displayed at the machine, along with a simulated view of the process. The setter/operator needs a high level of skill to perform the process, however the knowledge base is broader compared to the older production machines where intimate knowledge of each machine was considered essential. These machines are often set and operated by the same person, where the operator will supervise a small number of machines (cell).