3. PRE-PROCESSING PHASE

Pre processing involves building a mathematical model of the part you want to analyse. There are a few ways to do this, but by far the most common method is to take a 3D CAD model of the part and break itdown into thousands of tiny pieces that are a regular shape, such as a cube or a pyramid, through a process called meshing. Each tiny piece is called an element, (hence 'finite elementanalysis') and the corners of the elements are called nodes [10].

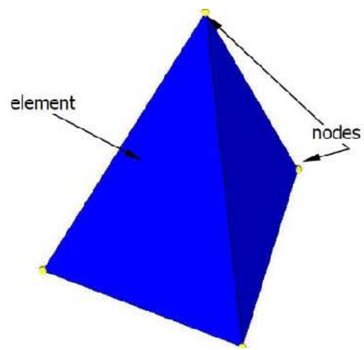


Fig. 29 Tetrahedral (pyramid) Element and Nodes [10]

So why split the model up like this? Well, not surprisingly, there's no mathematical formula to directly calculate the stress in a complex shape such as a suspension upright or a crankshaft.

There are however, formulas to calculate the stresses and displacements in a cube or pyramid when a load is applied to it. So the whole premise of FEA is to take a complex shape and break it down into tiny, regular shaped elements for which stress and strain can be calculated, then add all those results together to figure out the overall stress and strain within the part and the way it deforms due to the applied loads [10].

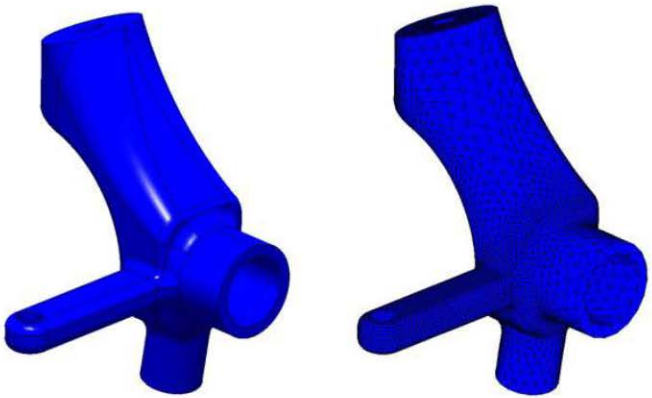


Fig. 30 Original CAD Model → Meshed FEA Model [10]

Pre processing software these days will take a CAD model and automatically mesh it with minimal input from the user. Element type, size, shape and quality does have a big effect on the accuracy of your results though, so it's not quite as simple as it sounds!

The more elements you have, the more accurate your results will be, but the analysis will take longer to run so it's a matter of finding a balance between accuracy and run time. Often a mesh is refined in areas of high stress or around complex shapes to increase the accuracy without increasing processing times.

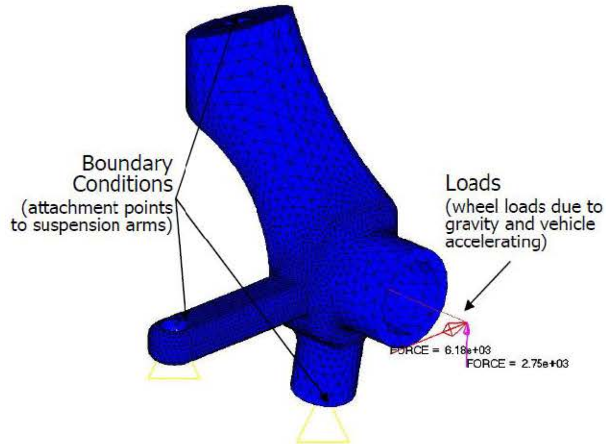


Fig. 31 Loads & Boundary Condition [10]

Once the model is meshed, material properties need to be definedand applied to the meshed part. These properties include the Young's modulus (a measure of material stiffness), its density, Poisson's ratio and more depending on the complexity of the analysis. The next step in the pre processing stage is to apply loads and boundary conditions to the model.

Loads are usually defined as forces acting on a certain point, but can also be torques, pressures, temperatures, or even a velocity or acceleration such as gravity. Boundary conditions are constraints that define how and where the part is held or bolted on, and are required to stop the part flying off into space when a force is applied. They basically tell the software which nodes aren't allowed to move during the analysis.

Once the model's been meshed, materials defined and loads and boundary conditions applied, you've now got a pre processed FEA model ready for solving [10].

3.1. Geometry modeling

3.1.1. General considerations about geometry modeling [40]

The components of a mechanical system, elastic deformable or elastic-plastic solids, have a variety of geometric shapes. In terms of the geometric shape of mechanical elements can be one-dimensional (1D), two-dimensional (2D), three-dimensional (3D) or combined.

The one-dimensional elements are objects wich having two dimensions smaller than the third. In practice, these elements come together, usually called as bars, beams, flexible, axles, shafts etc.

Depending on the shape of the axis, the bars can be straight (see Fig. 32, a, b) or curved (Fig. 32, c); from the point of view of the required applications, it can be traction-compression rods and / or bars required in bending and, after such changes in cross-section along the axis, the bars may be constant orbars variable section.

The bars can be divided as follows:

(i) depending on the form of the axis: straight bars (Fig. 32, a, b) or curved (Fig. 32, c);
(ii) from the point of view of applications: bars required tensile-compressive and / or flexural bars required;
(iii) according to the cross-section along the axis: bar with a constant or variable section.

Flexible elements - wires, bands, belts (Fig. 32, d) – wich have reduced bending stiffness, made of metallic or non-metallic type are considered having bar type geometry. Axles and straight shafts, as 1D-dimensional car elements, that support other elements in rotating machine are applied to bending and shear, respectively, torsion, bending, shear and tensile-compressive.

Geometrically, the 1D-elements are described using two elements: the axial curve and the cross-sectional shape and size.

Two-dimensional elements, as well as the constituent bodies having a much smaller size than the other two (Fig. 33,a), is found in practice that the tiles, membranes, coatings, dishes, etc.

Geometrically, these elements are described by defining the (i) shape and size of the median surface and (ii) width (Fig. 33,b).

Depending on the median surface, the plates can be flat or curved plates with single or double curvature. Commonly, the thickness of the plats used in practice is usually constant. For the small values of the thickness, the plats are called membranes.

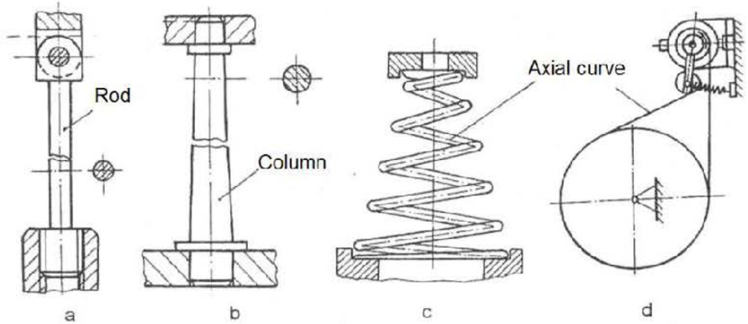


Fig. 32 One-dimensional elements [40]

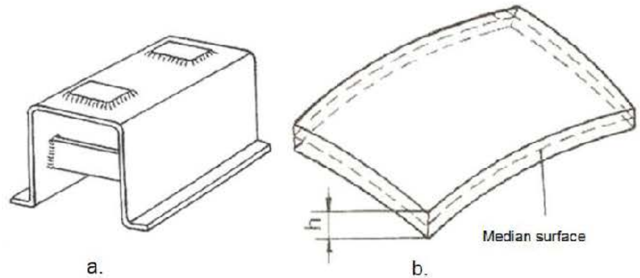


Fig. 33 Two-dimensional elements [40]

Three-dimensional elements, also referred to as solid bodies or blocks, have three dimensions about the same size and can not be reduced in any of the forms described above (Fig. 34,a).

The combined elements have geometry domain composed by two or more parties who framed the structures in the groups above presented. For example, Fig. 34,b shows the structure of a body composed of two sub-domains, two-dimensional and three-dimensional.

The concept of mechanical subassembly is primary, has general character, and is used to identify and study the smallest components or parts thereof. Analytical calculation methods of mechanical assemblies patterns associated to the elements are the same methods used in the strength of materials, characterized by increased levels of idealization and simplicity, in terms of the final design relationships.

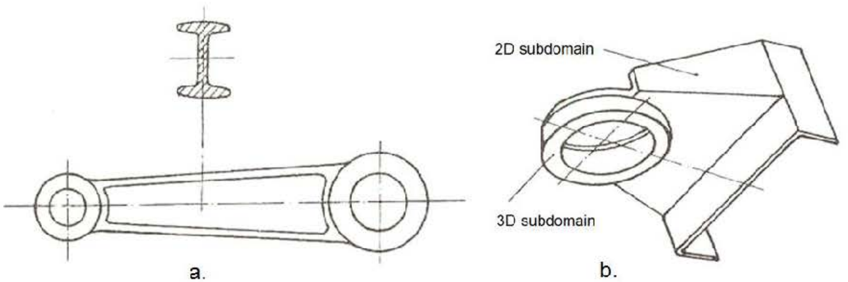


Fig. 34 Three-dimensional elements [40]

In the field of design, to analyze the components of a mechanical system (trusses, frames, platforms, tanks, machinery or equipment, etc..) those can be considered as one-dimensional geometric shapes, two-dimensional or three-dimensional, often cases lead to practical combinations thereof.

The modeled element or the subassembly in order to study based on FEM may be referred structure. In computer aided design, a structure described oneself using basic geometric entities: point, line, area and / or volume, relative to one or more previously defined coordinate systems. In addition, structures of analysis models, in particular mechanical systems, may include the idealized elements (typically, the rigid) provided by the used application software.

In order to geometric define the domains and subdomains of a structure, to prepare the finite element analysis model, you can use a Cartesian coordinate system (**Fig. 35**,a), cylindrical (**Fig. 35**, b) and / or spherical (**Fig. 35**, c). Each of these coordinate systems, depending of the geometric configuration of the structure domain, can be global reference system, to which the whole area of the problem, and a local reference system associated with each of a sub-domain thereof.

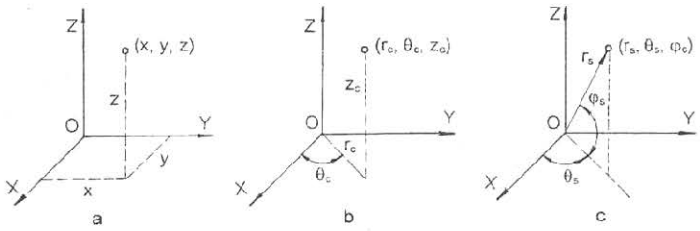


Fig. 35 Coordinate systems [40]

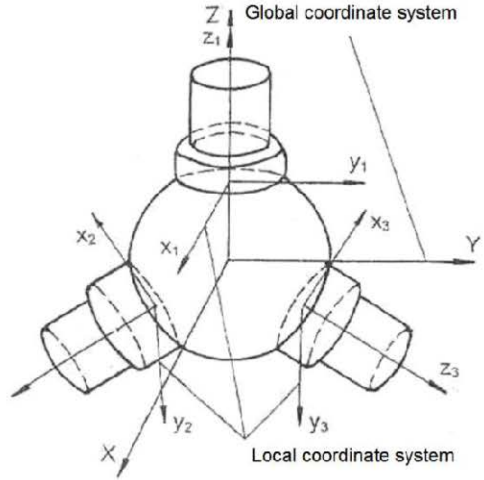


Fig. 36 Local and global coordinate systems [40]

In the example of Fig. 36, the Cartesian coordinate system is a global reference system, and the third cylindrical coordinate system are local system of reference.

Most FEM based application software have preprocessors that contain modules for geometric modeling. In addition, these programs are able to import geometric models from other programs mainly specialized in geometric modeling (AUTOCAD, EUCLID, ProEngineering etc.)

Geometric modeling of the structure domain using advanced programs is performed using basic geometric entities like lines, surface and / or volume that can be divided forming finite element sets. In order to identify, for finite element modeling, basic geometric entities have the following benchmarks: vertices, edges, and faces.

The structure domains and subdomains are generated by assembling the elementary entities using the commands from software library used. The following are common primary elementary entities: lines through points, arcs, circles, helix, involute, conic (ellipse, parabola, hyperbola) for unidimensional domanin; surfaces by dots or lines for surface domains; volume by points, lines or areas for volume domains.

Computer-aided geometric modeling of complex domains, based on elementary entities primary, secondary implies elementary entities which are obtained by additional operations (intersection, copy, offset, extrusion and so on).

Computer-aided geometric modeling of complex domains, based on primary elementary entities, implies the secondary elementary entities which are obtained by additional operations (intersection, copy, offset, extrusion and so on).

3.1.2. Basic concepts about 3D modeling

In 3D computer graphics, 3D modeling is the process of developing a mathematical representation of any three-dimensional surface of object (either inanimate or living) via specialized software. The product is called a 3D model. It can be displayed as a two-dimensional image through a process called 3D rendering or used in a computer simulation of physical phenomena. The model can also be physically created using 3D printing devices.

Models may be created automatically or manually. The manual modeling process of preparing geometric data for 3D computer graphics is similar to plastic arts such as sculpting. Recently, new concepts in 3D modeling have started to emerge. Recently, a new technology departing from the traditional techniques starts to emerge, such as Curve Controlled Modeling that emphasizes the modeling of the movement of a 3D object instead of the traditional modeling of the static shape [42].

3D models represent a 3D object using a collection of points in 3D space, connected by various geometric entities such as triangles, lines, curved surfaces, etc. Being a collection of data (points and other information), 3D models can be created by hand, algorithmically (procedural modeling), or scanned. 3D models are widely used anywhere in 3D graphics. Actually, their use predates the widespread use of 3D graphics on personal computers. Many computer games used pre-rendered images of 3D models as sprites before computers could render them in real-time [42].

Almost all 3D models can be divided into two categories.

- **Solid** - These models define the volume of the object they represent (like a rock). These are more realistic, but more difficult to build. Solid models are mostly used for nonvisual simulations such as medical and engineering simulations, for CAD and specialized visual applications such as ray tracing and constructive solid geometry.
- **Shell/boundary** - these models represent the surface, e.g. the boundary of the object, not its volume (like an infinitesimally thin eggshell). These are easier to work with than solid models. Almost all visual models used in games and film are shell models.

Because the appearance of an object depends largely on the exterior of the object, boundary representations are common in computer graphics. Two dimensional surfaces are a good analogy for the objects used in graphics, though quite often these objects are non-manifold. Since surfaces are not finite, a discrete digital approximation is required: polygonal meshes (and to a lesser extent subdivision surfaces) are by far the most common representation, although point-based representations have been gaining some popularity in recent years. Level sets are a useful representation for deforming surfaces which undergo many topological changes such as fluids.

The process of transforming representations of objects, such as the middle point coordinate of a sphere and a point on its circumference into a polygon representation of a sphere, is called tessellation. This step is used in polygon-based rendering, where objects are broken down from abstract representations ("primitives") such as spheres, cones etc., to so-called meshes, which are nets of interconnected triangles.

Meshes of triangles (instead of e.g. squares) are popular as they have proven to be easy to render using scanline rendering. Polygon representations are not used in all rendering techniques, and in these cases the tessellation step is not included in the transition from abstract representation to rendered scene [19].

There are three popular ways to represent a model:

- **Polygonal modeling** - Points in 3D space, called vertices, are connected by line segments to form a polygonal mesh. The vast majority of 3D models today are built as textured polygonal models, because they are flexible and because computers can render them so quickly. However, polygons are planar and can only approximate curved surfaces using many polygons.
- **Curve modeling** - Surfaces are defined by curves, which are influenced by weighted control points. The curve follows (but does not necessarily interpolate) the points. Increasing the weight for a point will pull the curve closer to that point. Curve types include nonuniform rational B-spline (NURBS), splines, patches and geometric primitives

- **Digital sculpting** - Still a fairly new method of modeling, 3D sculpting has become very popular in the few years it has been around. There are currently 3 types of digital sculpting: Displacement, which is the most widely used among applications at this moment, volumetric and dynamic tessellation. Displacement uses a dense model (often generated by Subdivision surfaces of a polygon control mesh) and stores new locations for the vertex positions through use of a 32bit image map that stores the adjusted locations. Volumetric which is based loosely on Voxels has similar capabilities as displacement but does not suffer from polygon stretching when there are not enough polygons in a region to achieve a deformation. Dynamic tesselation is similar to Voxel but divides the surface using triangulation to maintain a smooth surface and allow finer details. These methods allow for very artistic exploration as the model will have a new topology created over it once the models form and possibly details have been sculpted. The new mesh will usually have the original high resolution mesh information transferred into displacement data or normal map data if for a game engine.

The modeling stage consists of shaping individual objects that are later used in the scene. There are a number of modeling techniques, including:

- constructive solid geometry
- implicit surfaces
- subdivision surfaces

Modeling can be performed by means of a dedicated program (e.g., AutoCAD, ProEngineer, Unigraphics) or an application component or some scene description language. In some cases, there is no strict distinction between these phases; in such cases modeling is just part of the scene creation process (this is the case, for example, with Caligari trueSpace and Realsoft 3D).

Complex materials such as blowing sand, clouds, and liquid sprays are modeled with particle systems, and are a mass of 3D coordinates which have either points, polygons, texture splats, or sprites assigned to them [19].

The beginnings of computer designing in applications of CAD (Computer Aided Design) were based on 2D modelling of solids: machine-made elements, constructions, standard and non standard elements. 2D projections of a model are directly suitable for making the technical documentation: working and assembling drawings. However the main difficulties of such approach of designing aided by a computer were caused by a procedure of making 3D model out of already existing 2D projection in full display. Because of such difficulties, we were often obliged to hide or suspend certain displays on 2D models (such as dimensions and specific details) in order to do a process of third dimension extrusion. Such approaches were often the cause of technical mistakes and making of an unforeseeable time for their noticing, correction, removing, or at the worst case working out the existence of such a model.

When a new approach of solid products modelling appeared in CAD/CAM/CAE software family, a reversed procedure of modelling was created. First you create 3D model (which is a model that can be physically accomplished) out of which 2D projection is automatically created for the needs of technical documentation where there is no possibility for a mistake to appear and unnecessary losing of time and at the same time each change in 3D model is automatically reflected on all the applications derived from the model and there is no need for data interpretation.

Fig. 37 3D Assembly drawing [42]

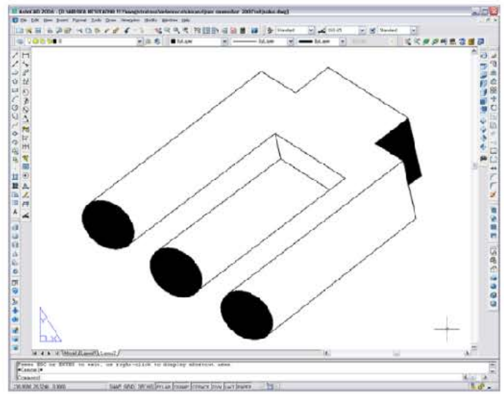


Fig. 38 Ambigous 3D model [42]

Therefore, when there is a need for certain modification of a project, for example an assembly (Fig. 37) it is possible to make corrections without returning to the beginning and to each model (part) individually, but one can directly make the corrections on the assembly itself [42]. Changing or defining of dimensions and other attributes of an element can be done at any moment but one should keep in mind that such changes are automatically transmitted to the whole model.

Solid modelling of a created computer model contains all the information of a real object. Those are models with cubic capacity which can have mass and inertia if specific solidity of the material is defined in advance. The important difference of this kind of modelling in relation to surface modelling is that orifices and holes in the model of solid automatically create new surface so that one can precisely determine which side of surface represents solid material [42].

At the very access to software for 3D modelling of solids such as: ProENGINEER WILDFIRE (which is applied in this work); Solid Works; Solid Edge; CATIA; Autodesk Inventor, etc. in which one wants to create a model by means of constructive elements he or she has to keep in mind that a model which cannot be physically accomplished also cannot be projected in the applications mentioned above or some of them can be projected but one should pay attention to estimation of its probability to be manufactured in machine made process.

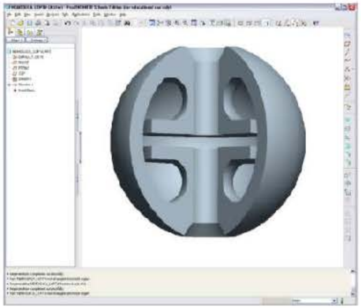


Fig. 39 3D model [42]

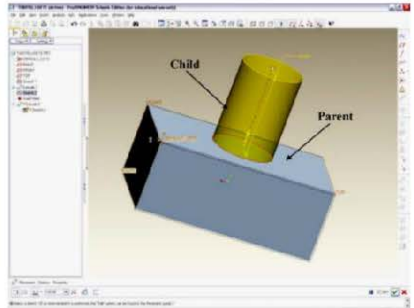


Fig. 40 Parent / child relationship[42]

The displayed model in (Fig. 38) represents an object which resembles three-sided spatial object but on the occasion of solid modelling it is not possible to create such a model which is ambiguous and cannot exist physically. However such a model is very simply suitable for creation in 2D, wire and surface models.

Let us observe the part of a model in (Fig. 39) which can be physically accomplished and can be modelled by means of computer as a solid product but there is a question imposed whether the machines can make such orifices inside the displayed model the estimation of which is the authority of a person who designed this model [42].

Constructive elements represent basis for solid products modelling and they possess reciprocal relations which are practically called Parent/Child relationship. Constructive element "parent" is the element that represents the base, that is to say a reference for the creation of a new element which will have the characteristic "child" because of the subordination which is created by its existence.

Parents and children can be surfaces of some models, planes, angles, axes, points, and the like, which are reciprocally subordinated in such a way that if, for example one deletes the element "parent", its elements "children" will also disappear. At solid product modelling it is very important to be careful about these relationships and principles for their creation because on the contrary a seemingly simple constructive problem can turn into an extremely complicated one especially at the moments when one changes or deletes some elements which possess parent/child relationship [42].

The applications mentioned above for 3D modelling mainly possess special functions for the work with parent/child relationships which have to be studied well before engaging in modification of a project. In (Fig. 40) one can observe the relation of referential surface of prismatic part which represents the element "parent" on which there is a constructive element created of the circle "child" for the extrusion of cylindrical part.

3.1.3. Samples of the various 3D modeling software

The following is the sequence of operations for drawing 3D model of a castor support (Fig. 41)

3.1.3.1. 3D modelling in CATIA

Step 1: Activating the generation of solids and set the unit of length (Fig. 42)

Start → Mechanical Design → Part Design → **New part**: New part name: Suport.

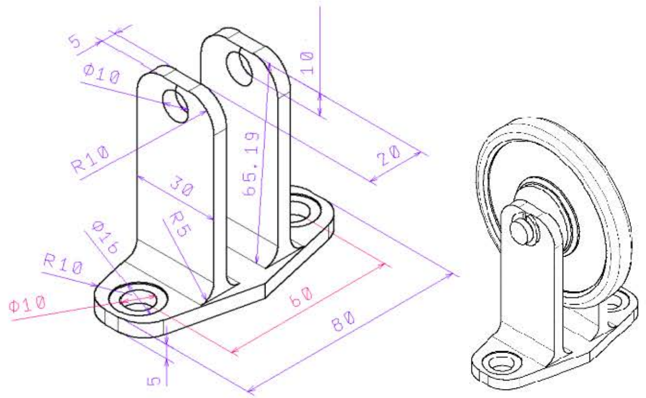


Fig. 41 Fixed wheel support

Tools → Options …→ **Options**: Parameters and Measure; Units; Length, Milimeter (mm); ↵ OK.

Step 2: Generating reference sketch. Will choose to draw a half of part, considering the symmetry of the piece (Fig. 43).

 (Sketcher) → xy plane →  (Profile) → [base contour is drawn] →  (Constraint) [quotas were introduced successively by selecting lines using Ctrl key to select multiple] →  (Exit Workbench).

Fig. 42 Choosing Part Design module

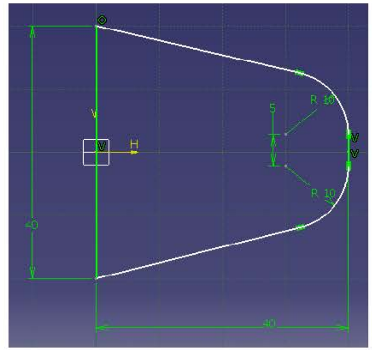


Fig. 43 Reference sketch

Step 3: Solid model generation (Fig. 44)

 (Pad) → **Pad Definition**: Selection [selecting Sketch.1], Lenght = 5 mm, ↵OK.

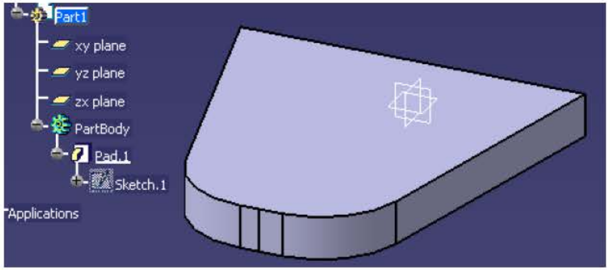


Fig. 44 Pad definition

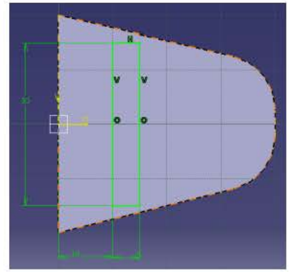


Fig. 45 Creating vertical zone I

Step 4: Creating vertical zone (Fig. 45, Fig. 46)

(Sketcher) → [selection upper surface of the part] → (Profile) → [base contour is drawn] → (Constraint) [quotas were introduced successively by selecting lines using Ctrl key to select multiple] → (Exit Workbench).

(Pad) → **Pad Definition**: Selection [selecting Sketch.2], Lenght = 65 mm, ↲OK;

(Edge Fillet) → **Edge Fillet Definition**: Radius=10 mm, Object(s) to fillet: selection Pad.2/TgtEdge.1 ↲OK (repeat for the both edges and for connecting area with the base part, using radius = 5 mm) (Fig. 47).

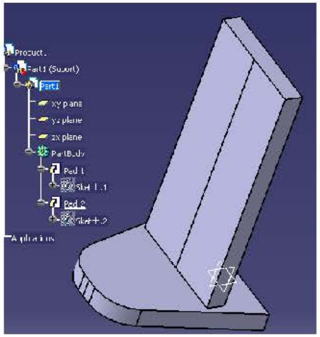


Fig. 46 Creating vertical zone II

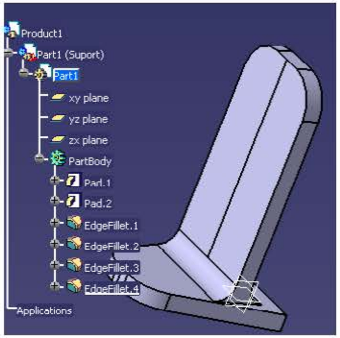


Fig. 47 Edge fillet

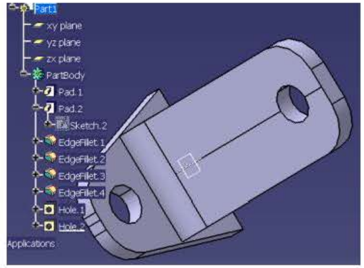


Fig. 48 Drawing holes

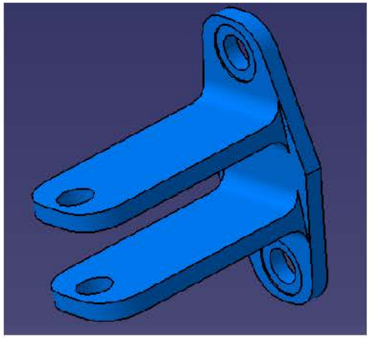


Fig. 49 Completion phase

Step 5: Drawing holes (**Fig. 48**)

 (Hole) → [selection surface on Pad.2] → (Point by Using Coordinates) **Hole Definition: Extension tab** - Up to Next, Diameter = 10 mm,  Normal to surface,  Positioning Sketch (hole is positioned and quoted using Constraints command); Type tab – Counterbored, Diameter = 16 mm, Depth = 0,5 mm ↵OK

Step 6: Completion phase (**Fig. 49**)

 (Mirror) → **Mirror Definition:** Mirroring element: yz plane, Object to mirror: Curent solid; (Edge Fillet) → **Edge Fillet:** Definition: Radius=20 mm, Object(s) to fillet: selection edges remaining after mirroring ↵OK; in the tree, right click on PartBody→ Properties → Graphic tab → select other colour in Fill Colour ↵OK

3.1.3.2. Assembling in CATIA

Separately, were built the other two components of the assembly: the wheel (Fig. 50) the shaft (Fig. 51), and the ring (Fig. 52).

Fig. 50 The wheel

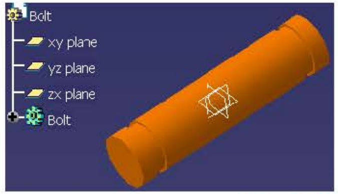


Fig. 51 The shaft

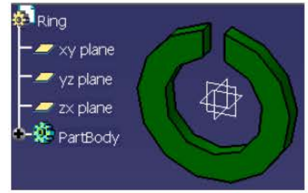


Fig. 52 The ring

Step 1: Activating the assembly module.

Start → Mechanical Design → Assembly Design

Open all the parts that you built (Suport, wheel, shaft and ring).

Step 2: Inserting components (Fig. 53)

Select on the tree Product.1 → Go to the Suport file (Window → Suport.CatPart) → select PartBody in the tree → Copy → return in Product.1 file → Paste.

Do the same with all the components; for the ring, copy it twice.

Step 3: Positioning components

Holding the Ctrl key, select the folowing cylinders, twos: wheel hole, ring interior hole, shaft and suport holes (Fig. 54) →  (Coincident constraint) (Fig. 55).

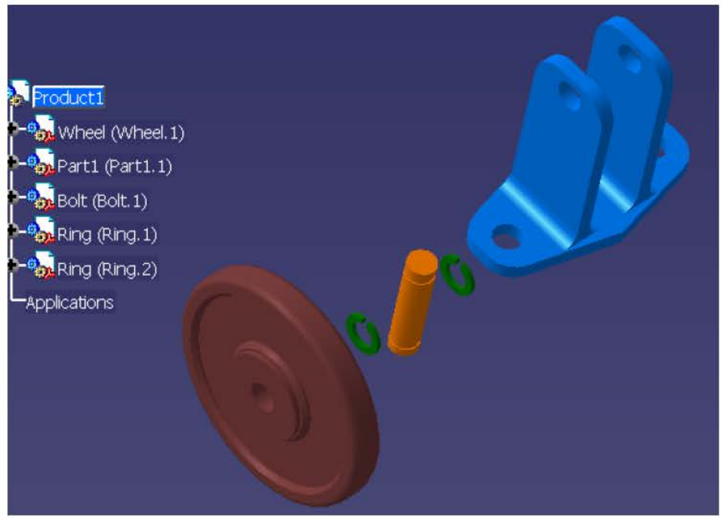


Fig. 53 Inserting and positioning components

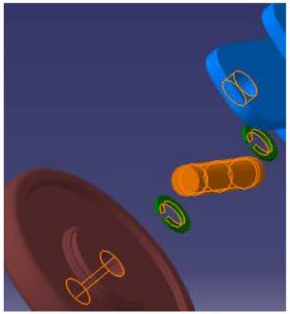


Fig. 54 Importing components

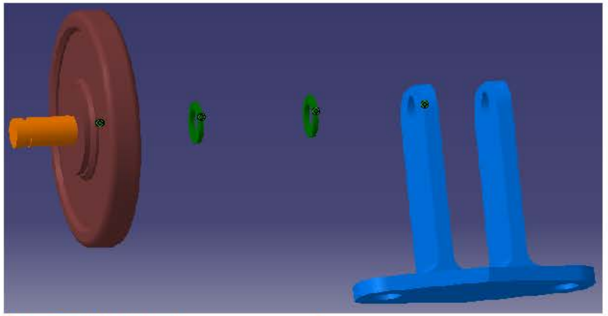


Fig. 55 Positioning components

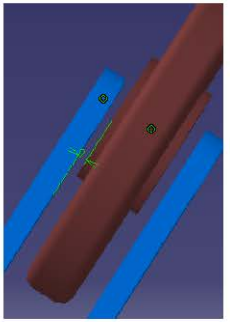


Fig. 56 Constraint the elements

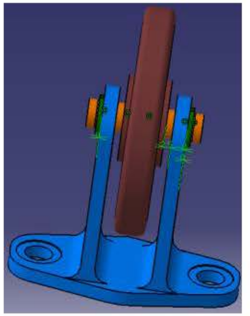


Fig. 57 Finishing the assembly

Selecting the interior surface of suport and exterior surface of planar wheel hub →  (Offset Constraint) → **Constraint Properties:** Orientation: Opposite, Offset = 2,5 mm, ↵OK (Fig. 56). In the same way, place all the pieces (Fig. 57).

3.1.3.3. 3D modelling in ANSYS

This tutorial is intended to demonstrate the DesignModeler modelling tools: how to use the 3d modelling tools such as revolve, extrude loft etc. The DesignModeler comes with ANSYS. While working in industry you will be travelling here and there and one of the obstacles you need to get around regularly every now and then is that at a certain company they use CATIA as a modelling software while at another they use AUTOCAD then at another they use SolidWorks at another they use ….. etc.

ANSYS Workbench is a CAD-neutral environment that supports bidirectional, direct, and associative interfaces with CAD systems. You can manipulate existing native CAD geometry directly without translation to intermediate geometry formats. Via plug-ins, the associative interface allows you to make parametric changes from either a CAD system or from within ANSYS Workbench.

The DesignModeler solves this problem it's modelling software provided with the ANSYS package [2]. The DesignModeler application is designed to be used as a geometry editor of existing CAD models. The DesignModeler application is a parametric feature-based solid modeler designed so that you can intuitively and quickly begin drawing 2D sketches, modeling 3D parts, or uploading 3D CAD models for engineering analysis preprocessing.

If you have never used a parametric solid modeler, you will find the DesignModeler application easy to learn and use. If you are an experienced user in parametric modeling, the DesignModeler application offers you the functionality and power you need to convert 2D sketches of lines, arcs, and splines into 3D models.

The DesignModeler application's interface is similar to that of most other feature-based modelers. The program displays toolbars along the top of the screen [6].

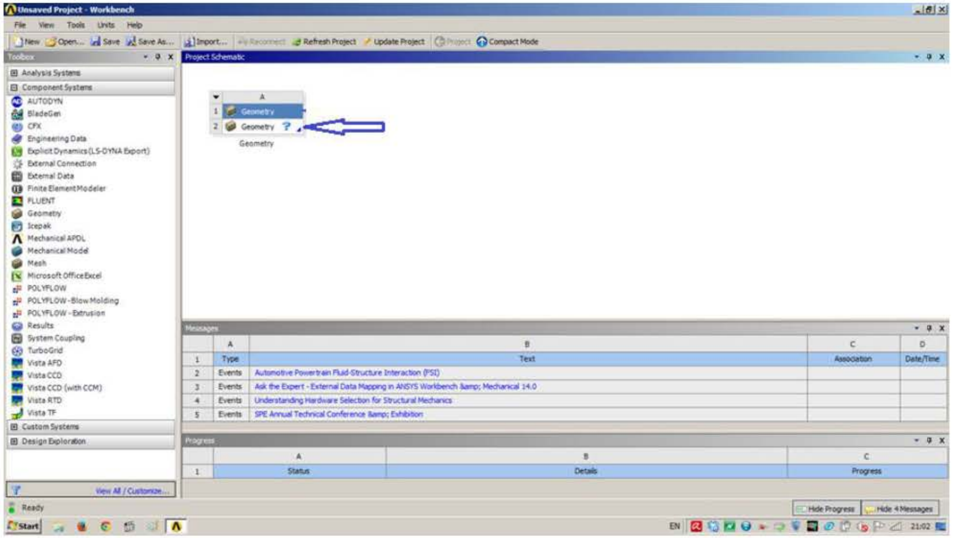


Fig. 58 The main window of Ansys 14.0; starting the geometry

The DesignModeler application features two basic modes of operation: 2D Sketching and 3D Modeling. The DesignModeler application is a parametric feature-based modeler. Its modeling paradigm is to sketch 2D profiles and use them to generate features. In CAD systems, features are collections of geometric shapes with which you add or cut material from a model. In the DesignModeler application, you can also use features to slice a model into separate bodies for improved mesh generation or to imprint faces for patch loading. More generally, in the DesignModeler application you can apply features to the task of enhancing your models for the purpose of engineering simulation.

Because the DesignModeler application is a feature-based modeler, the features shown in the feature Tree Outline list all of the operations used to create the model. This feature list represents the model's history. Features may be modified and the model rebuilt to reflect your changes. Features may also be suppressed, deleted, or even inserted into the middle of the feature list.

A sketch is always required at the start of creating a new model. However not all features, such as Blend and Chamfer, require you to create sketches. Some features, such as Extrude or Sweep, require you to create sketches prior to their definition.

During this tutorial a simple geometry is used, the objective of that is that the student masters the steps to get to run a simple simulation, once that's done the student can model any kind of geometry he sees necessary for his studied case.

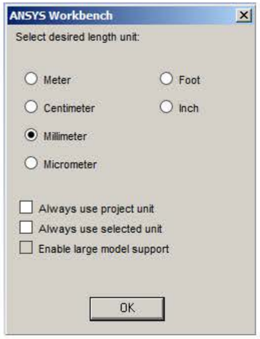


Fig. 59 Units selection

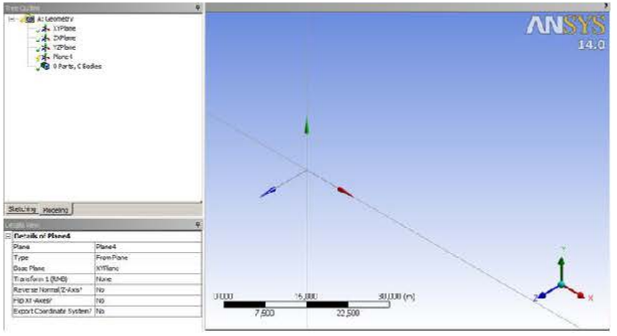


Fig. 60 3D Space for drawing

Solid modeling can be accomplished in a number of ways, and one favorite method involves starting with a two-dimensional shape and manipulating it to create a solid. That is the approach we will use for many of the object models created in this book.

Step 1: Launch ANSYS, by going to the start-up menu and double clicking on workbench file in the ANSYS 14.0 folder, Go to Toolbox / Component Systems and double click.

Step 2: select Milimeter as the dimension unit by double click on Geometry (in Project Schematic window – see Fig. 58, Fig. 59). Units selection depends on what kind of units is preferred by the manufacturer, if you were dealing with a customer in United States he will be using foot while if you were dealing with a customer in the united kingdom he will be using inch.

Step 3: click on the icon  that has the Cartesian coordinate's icon. Once it's clicked the user can see where is the plotting plane (Fig. 60).

Step 4: Looking at the window you will see the Cartesian coordinates system is shown on the screen with a blue arrow referring to the z axis, green arrow referring to the y axis, red arrow referring to the x axis. A note to the user that 2d sketching is conducted on the specified xy plane as long as the user specifies some other plotting plane (xz plane, yz plane). The user can access the modelling tools by pressing on the Sketching tab or he can access the modelling planes by going to Modelling. You can choose other viewing options provided under views (Fig. 61).

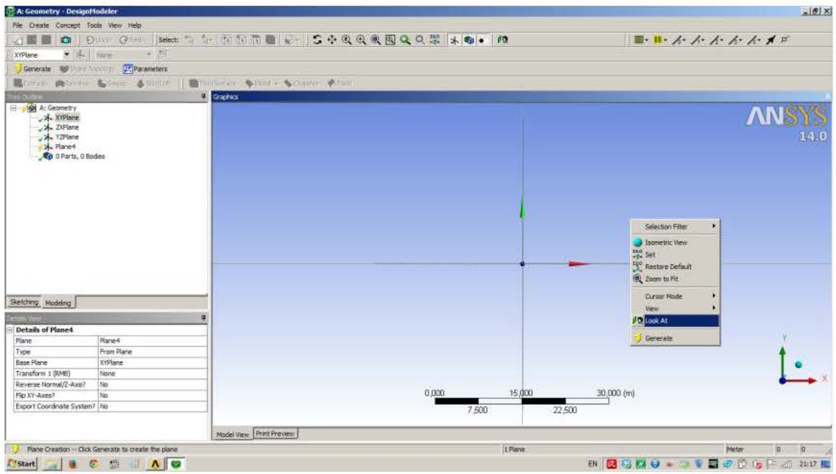


Fig. 61 Viewing options

Step 5: Select the Line and start sketching the base profile, the line command is done by pressing twice, once you have drawn the first line, press again on the last plotted point and continue till you plot all the profile, remember to make sure that the profile contour is closed, now that the 2d sketch is done press on the extrude icon. You can specify the height of the extrude, editing the height is shown in step 6.

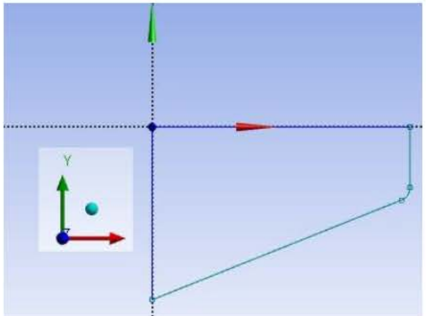


Fig. 62 Sketcher

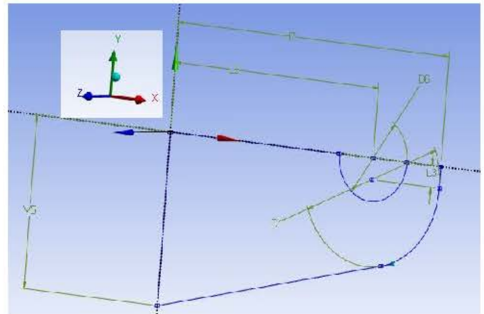


Fig. 63 Use dimensions commands

When you finish the sketch, right click and select closed end.

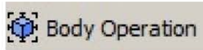
Sketch View → use Constraints if necessary to align them.

Dimensions → place the dimensions as shown below. To obtain the following view, first get the isometric view to figure out the orientation by RMB on the model window → View → Isometric View. Now, RMB → View → Top View. Remember Dimensions → Display → check Value and uncheck Name to display numerical values.Now the picture changes from a dark set of lines to this lighter outline. Click on the ball to place yourself in ISO view and on the Modeling button to close the sketching menu and return to the tree diagram.

Step 6: Hit the Extrude button 

. Going to the Details view and changing the value from 1 to 12 will change the extrude height of the profile. Then press 

Generate, forgetting to press the generate button will not get the solid generated. Try selecting different options provided in the Details View to see what effect it has on the model (Fig. 64). Note the color of the object. We can still change the dimensions and then hit generate and it will recreate the object in the new length. Note how it says we have one body and one part. If we want to create another part without interacting with this one, we can "Freeze" this part by selecting Tools/Freeze. We can now make other sketches and extrude and generate to create all the pieces. Once we have an object we like, we can mirror it or move it by generating additional planes.

Step 7: Creating horizontal body. Create → 

(Body Operation) → pick the body → Apply in Details View → click Mirror Plane to bring up Apply/Cancel → pick XZ Plane in Tree Outline → Generate. See mirrored part appears as. In order to draw a counterbored hole, draw a circle on a superior face of surface, with axis coincidence with hole axis with dimension D = 16 mm.Using option from Details of Extrudes → Operation → Cut Material, extrude with Depth = 0,5 mm. The extruded profile should look something similar to this (Fig. 65).

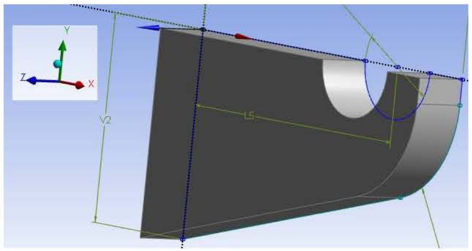


Fig. 64 Changing the dimensions

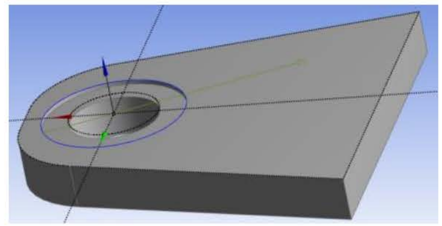
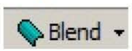
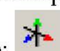
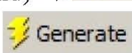


Fig. 65 Using mirror comand

Step 8: Creating vertical zone. Now that the solid has been generated you can click on Modelling the you can select what plane to work on click on the icon that has a box with a green square surface and click on the top surface of the extruded profile. You have to draw a rectangle (Fig. 66). Use Dimensions and Constraints facilities to fix it, after that you have to extrude (Fig. 67). The sharp edges should be rounded with 

(Blend) → 

(Fixed radius) command and you have to draw a hole (Fig. 68):  (New Plane) → Selected the lateral surface → 

(Generate) → Draw a circle → Constraints → Dimensions D=10 mm → 

(Extrude) → Cut material.

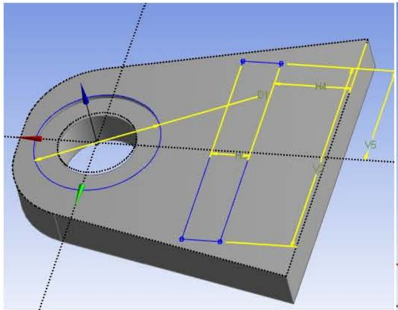
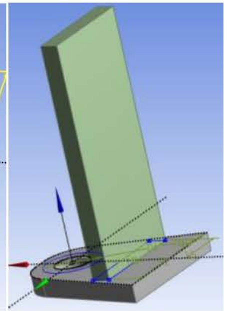
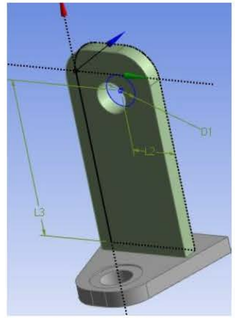
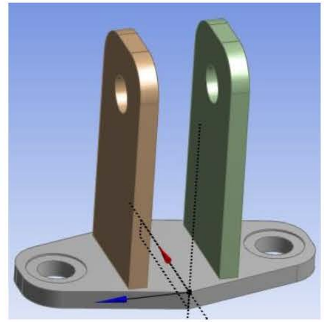
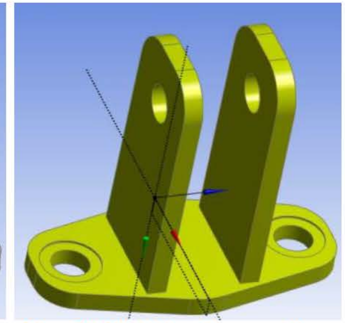
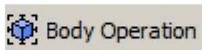






Fig. 66 Drawing a rectangle **Fig. 67 Extruding** **Fig. 68 Fillet** **Fig. 69 Part construction in ANSYS** **Fig. 70 Finishing the part in ANSYS**

Step 9: Finishing the piece. Selecting from the Tree Outline Solid 1 and Solid 2, go to Create → 

(Body Operation) → Type: Mirror, Mirror Plane: YZ, → 

(Generate) (Fig. 69). Selexct Solid 1, Solid 2, Solid 3 → Form a new part (**Fig. 70**).

3.1.3.4. Assembly in ANSYS

Since drawing module is not very developed, it is recommended to import assemblies made with other advanced 3D modeling software. New imported parts are shown in (Fig. 71).

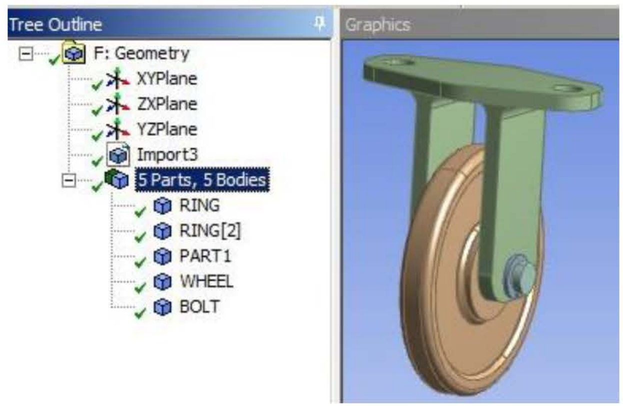


Fig. 71 Finishing assembly in ANSYS

3.1.3.5. Importing 3D model in Z88 Aurora

This software does not have the ability to make a new 3D model. For this reason, the elements which will studied will be made in other programs and then will import the Z88.

Step 1: Launching a New Project Folder

- Create a new folder
- Enter folder name „*Name*"
- Confirm (Return) and *double click* (left mouse button) to activate folder
- Click OK to confirm

The input mask disappears, you can start the compilation of the computation model. For further use, the project folder can be put into the quick access! (⇨ Add).

Step 2: Opening a Project Folder.

Select a project folder to open.

Select Project information, all relevant data about the project folder are shown

Click "OK" to confirm. The project is displayed in the work area.

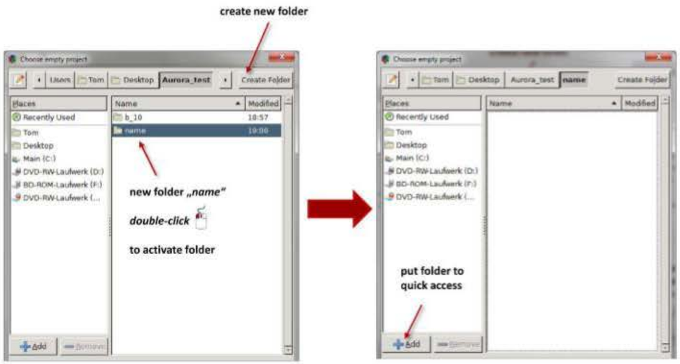


Fig. 72 Launching a new project folder

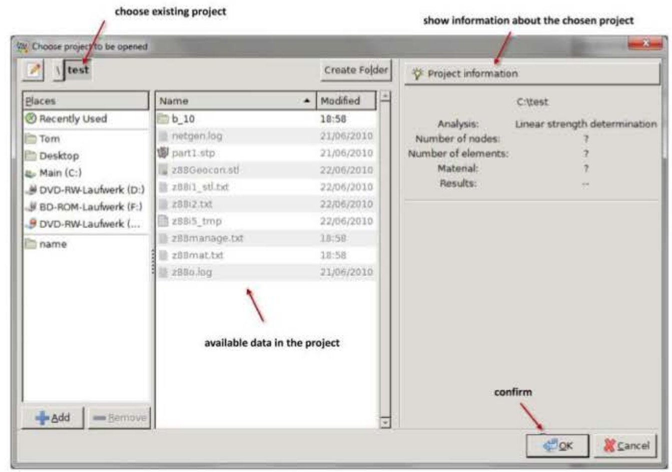
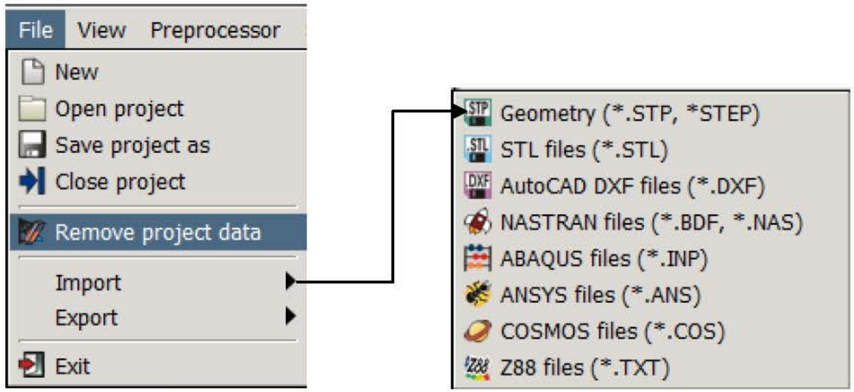


Fig. 73 Opening an existing project folder

Step 3. Import a geometry (Fig. 74)

File → Import → Choose file type (*.STP, *.STL, *.DXF, *.NAS, *.INP, *.ANS),



or use icons from toolbar:  .

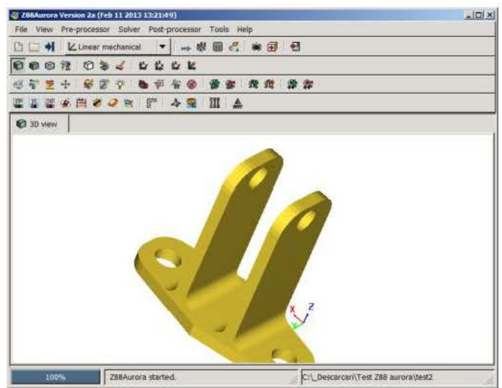


Fig. 74 3D model imported from *.stp file

3.2. Assign material properties

From the point of view of the internal structure and implicitly the physical properties, materials can be homogeneous, with the same structure and physical properties in all points, or inhomogeneous, when this condition is not met. For homogeneous materials, physical parameters are constant related to spatial geometric variables.

Depending on the type of variation of physical properties in the vicinity of each point, there are isotropic materials, with properties independent of the direction to which it relates, or anisotropic, with properties dependent to the direction by which it is considered.

The properties of isotropy and homogeneity are not mutually conditioned. A solid body can be both homogeneous and isotropic, homogeneous and anisotropic, isotropic and inhomogeneous or even inhomogeneous and anisotropic [40].

3.2.1. Modeling the mechanical behavior of materials

Solid materials which after deformation (change in shape) under the action of loads return to the original shape and size are called elastic and those which do not comply with this condition are called plastics. In the case of mechanical system elements, usually the behavior of the material is elastic, or, sometimes, elasto-plastic. This occurred even at mechanical system elements that, in order to obtain the final form, technological processes based on plastic deformations cold or hot were used.

To determine the stress, deformation, thermal and displacement fields of a constant solid, deformable under the action of external loads (Fig. 18,a), it is necessary to know the laws of mechanical behavior of these materials.

The behavior of elastic materials can be linear, when the stress-strain dependencies are linear, or nonlinear, when these dependencies are linear. Assuming linearity, for any material, the variation of stress vector components (see Eq.5) between the initial state as a reference (a random point P, Fig. 18,a) and the final state (point P ') is described as a linear combination of the deformation vector components (see Eq.4), the relationship

$$[\delta] = [E]\ [\varepsilon], \qquad \text{Eq. 6}$$

in which,

$$[E]=\begin{bmatrix} E_{xxxx} & E_{xxyy} & E_{xxzz} & E_{xxyz} & E_{xxxz} & E_{xxxy} \\ E_{yyxx} & E_{yyyy} & E_{zzyy} & E_{yyyz} & E_{yyxz} & E_{yyxy} \\ E_{yyxy} & E_{zzyy} & E_{zzzz} & E_{zzyz} & E_{zzxz} & E_{zzxy} \\ E_{yzxx} & E_{yzyy} & E_{yzzz} & E_{yzyz} & E_{yzxz} & E_{yzxy} \\ E_{xzxx} & E_{xzyy} & E_{xzzz} & E_{xzyz} & E_{xzxz} & E_{xzxy} \\ E_{xyxx} & E_{xyyy} & E_{xyzz} & E_{xyyz} & E_{xyxz} & E_{xyxy} \end{bmatrix} \qquad \textbf{Eq. 7}$$

is the material elasticity matrix. Therefore, in theoretical cases, the stress and symmetric strain tensors (tangential stresses and angular deformations apply to the relations: $\tau_{ij} = \tau_{ji}$ and $\gamma_{ki} = \gamma_{ik}$), the material elasticity matrix contains 36 elements, called constants (modules) of elasticity. These constants characterize the material response to tri-orthogonal axial loads (traction, compression and torsion in both directions).

Considering the elasticity symmetry, the matrix (Eq. 7) takes the following shape

$$[E]=\begin{bmatrix} E_{xxxx} & E_{xxyy} & E_{xxzz} & E_{xxyz} & E_{xxxz} & E_{xxxy} \\ & E_{yyyy} & E_{zzyy} & E_{yyyz} & E_{yyxz} & E_{yyxy} \\ & & E_{zzzz} & E_{zzyz} & E_{zzxz} & E_{zzxy} \\ & & & E_{yzyz} & E_{yzxz} & E_{yzxy} \\ & \text{Simetric} & & & E_{xzxz} & E_{xzxy} \\ & & & & & E_{xyxy} \end{bmatrix} \qquad \textbf{Eq. 8}$$

Consequently, for the linear elastic material it's showing 21 virtually spring constants.

The materials consisting of parallel fibers embedded in a constitutive homogeneous mass is characterized by the symmetry to a normal plane in the direction of the fibers. In this case, the number of independent elastic constants is reduced to 13 and if OX is parallel with the material fibers and the symmetry plane is OZY, the elasticity matrix (Eq. 8) becomes

$$[E]=\begin{bmatrix} E_{xxxx} & E_{xxyy} & E_{xxzz} & E_{xxyz} & 0 & 0 \\ & E_{yyyy} & E_{zzyy} & E_{yyyz} & 0 & 0 \\ & & E_{zzzz} & E_{zzyz} & 0 & 0 \\ & & & E_{yzyz} & 0 & 0 \\ & \text{Simetric} & & & E_{xzxz} & E_{xzxy} \\ & & & & & E_{xyxy} \end{bmatrix}$$

Eq. 9

In the case of materials with two orthogonal symmetry planes, called orthotropic or orthogonally anisotropic, describing the linear elastic behavior of a material is made with 9 independent elastic coefficients, such as the non-zero elements of the elasticity matrix

$$[E]=\begin{bmatrix} E_{xxxx} & E_{xxyy} & E_{xxzz} & 0 & 0 & 0 \\ & E_{yyyy} & E_{zzyy} & 0 & 0 & 0 \\ & & E_{zzzz} & 0 & 0 & 0 \\ & & & E_{yzyz} & 0 & 0 \\ & \text{Simetric} & & & E_{xzxz} & 0 \\ & & & & & E_{xyxy} \end{bmatrix}$$

Eq. 10

In this case, implicitly, there is a third plane of elastic symmetry, orthogonal with the first two. The elastic constants corresponding to the attached tri-orthogonal coordinate straight system directions are called main elasticity constants. The determination of these elasticity constants is achieved by experimental tests on test tubes, taking into account the hypothesis of the inexistence of "coupling" between normal and specific shear stresses corresponding to the orthotropic axes.

Isotropic materials are characterized by some invariable types of behavior for every defining direction and, therefore, the elasticity matrix, with the same form, regardless of the direction of the reference axes of the coordinate system adopted in the practice of design has the following configuration:

$$[E]=\begin{bmatrix} \lambda+2\mu & \lambda & \lambda & 0 & 0 & 0 \\ & \lambda+2\mu & \lambda & 0 & 0 & 0 \\ & & \lambda+2\mu & 0 & 0 & 0 \\ & & & 2\mu & 0 & 0 \\ & \text{Simetric} & & & 2\mu & 0 \\ & & & & & 2\mu \end{bmatrix} \qquad \textbf{Eq. 11}$$

with the two independent elasticity coefficients defined by the relationships:

$$\lambda=\frac{\nu E}{(1+\nu)(1-2\nu)} \qquad \textbf{Eq. 12}$$

$$\mu=\frac{E}{2(1+\nu)} \qquad \textbf{Eq. 13}$$

where **E**, the longitudinal elasticity module and **ν**, the transverse contraction coefficient, are the technical elasticity constants of the material.

3.2.2. Modeling thermal behavior of materials

In many practical situations, the elements of mechanical systems are working at different temperatures of those considered as normal, as a consequence of the existence of internal or external heat sources. As a result, in the structures of these elements occur thermal stress and strains that overlap with the mechanical ones which appear as consequence of requests during functioning..

To consider the finite element analysis of thermal effects, it is necessary to know the thermal characteristics of the structure materials to be analyzed. Usually, for thermomechanical analyses of structures it is requisite that for the materials from which they are made, to know the specific heat and the conductivity coefficients and thermal expansion related to the directions of a tri-orthogonal straight coordinate system.

The specific heat is the characteristic parameter which quantifies the heat storage capacity of the material and it is measured in J / kg K.

The coefficient of thermal conductivity, measured in W/mK is a material constant variable with temperature. This variation, generally, is quasi-linear and can be described with the relationship [40]

$$\lambda = \lambda_0 (1 - bT) \qquad \textbf{Eq. 14}$$

where λ_0 is the coefficient of thermal conductivity at the temperature of 0° C and **b** - a experimentally determined constant.

The linear thermal expansion coefficient α, expressed in K^{-1}, quantifies the distortion ability of the material under the action of the thermal fields, which also varies with temperature.

3.3. Meshing (discretization) – types of finite elements 3D, 2D, 1D

3.3.1. Discretization [17]

Mathematical modeling is a simplifying step. But models of physical systems are not necessarily simple to solve. They often involve coupled partial differential equations in space and time subject to boundary and/or interface conditions. Such models have an infinite number of degrees of freedom.

3.3.1.1. Analytical or Numerical

At this point one faces the choice of going for analytical or numerical solutions. Analytical solutions, also called "closed form solutions," are more intellectually satisfying, particularly if they apply to a wide class of problems, so that particular instances may be obtained by substituting the values of free parameters. Unfortunately they tend to be restricted to regular geometries and simple boundary conditions. Moreover some closed-form solutions, expressed for example as inverses of integral transforms, may have to be anyway numerically evaluated to be useful.

Most problems faced by the engineer either do not yield to analytical treatment or doing so would require a disproportionate amount of effort. The practical way out is numerical simulation. Here is where finite element methods enter the scene.

To make numerical simulations practical it is necessary to reduce the number of degrees of freedom to a finite number. The reduction is called discretization. The product of the discretization process is the discrete model. For complex engineering systems this model is the product of a multilevel decomposition.

Discretization can proceed in space dimensions as well as in the time dimension. Because the present book deals with static problems, we need not consider the time dimension and are free to focus on spatial discretization.

3.3.1.2. Error Sources and Approximation

Fig. 16 conveys graphically that each simulation step introduces a source of error. In engineering practice modeling errors are by far the most important. But they are difficult and expensive to evaluate, because model validation requires access to and comparison with experimental results. These may be either scarce, or unavailable in the case of a new product in the design stage.

Next in order of importance is the discretization error. Even if solution errors are ignored — and usually they can—the computed solution of the discrete model is in general only an approximation in some sense to the exact solution of the mathematical model. A quantitative measurement of this discrepancy is called the discretization error. The characterization and study of this error is addressed by a branch of numerical mathematics called approximation theory.

Intuitively one might suspect that the accuracy of the discrete model solution would improve as the number of degrees of freedom is increased, and that the discretization error goes to zero as that number goes to infinity. This loosely worded statement describes the convergence requirement of discrete approximations. One of the key goals of approximation theory is to make the statement as precise as it can be expected from a branch of mathematics

3.3.1.3. Other Discretization Methods

The most popular discretization techniques in structural mechanics are finite element methods and boundary element methods. The *finite element method* (FEM) is by far the most widely used.

The *boundary element method* (BEM) has gained in popularity for special types of problems, particularly those involving infinite domains, but remains a distant second, and seems to have reached its natural limits.

In non-structural application areas such as fluid mechanics and electromagnetics, the finite element method is gradually making up ground but faces stiff competition from both the classical and energybased finite difference methods.

Finite difference and *finite volume* methods are particularly well entrenched in computational fluid dynamics spanning moderate to high Reynolds numbers.

3.3.2. The Finite Elements

The finite element method (FEM) is the dominant discretization technique in structural mechanics. The FEM can be interpreted from either a physical or mathematical viewpoint.

The basic concept in the physical FEM is the subdivision of the mathematical model into disjoint (non-overlapping) components of simple geometry called finite elements or elements for short. The response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function, or functions, at a set of nodal points. The response of the mathematical model is then considered to be approximated by that of the discrete model obtained by connecting or assembling the collection of all elements.

The disconnection-assembly concept occurs naturally when examining many artificial and natural systems. For example, it is easy to visualize an engine, bridge, building, airplane, or skeleton as fabricated from simpler components.

Unlike finite difference models, finite elements do not overlap in space. In the mathematical interpretation of the FEM, this property goes by the name disjoint support or local support.

3.3.3. The meshing procedure

Just like members in the truss example, one can take finite elements of any kind one at a time. Their local properties can be developed by considering them in isolation, as individual entities. This is the key to the modular programming of element libraries.

In the Direct Stiffness Method, elements are isolated by the disconnection and localization steps. The procedure involves the separation of elements from their neighbors by disconnecting the nodes, followed by referral of the element to a convenient local coordinate system. After that we can consider generic elements: a bar element, a beam element, and so on. From the standpoint of the computer implementation, it means that you can write one subroutine or module that constructs, by suitable parametrization, all elements of one type, instead of writing a new one for each element instance.

Following is a summary of the data associated with an individual finite element. This data is used in finite element programs to carry out element level calculations.

The numerical methods (including FEM) of physical phenomena analysis occurring in continuous geometric domains involve replacing them with idealized domains (approximate) assemblies of smaller domains, in the case of FEM called finite elements. The borders of finite elements consist of points (nodes), straight lines or curves (nodal lines) and/ or flat or random planes (nodal surfaces).

The choosing operation of the number of nodes and the type of line or surface while respecting continuities on a nodal level, with the scope of finite element modeling geometric domains, is called meshing. Fig. 75 shows, for example, the two-dimensional, D, meshed in triangular finite elements with straight side lines (Fig. 75, a) and curved lines sides (Fig. 75,b). In the first case, that of finite elements with straight side lines, the meshing error can be reduced by increasing the number of nodes and so, implicitly, the number of finite elements. In the second case, the meshing error can be decreased also thanks to the curved borders of the finite elements.

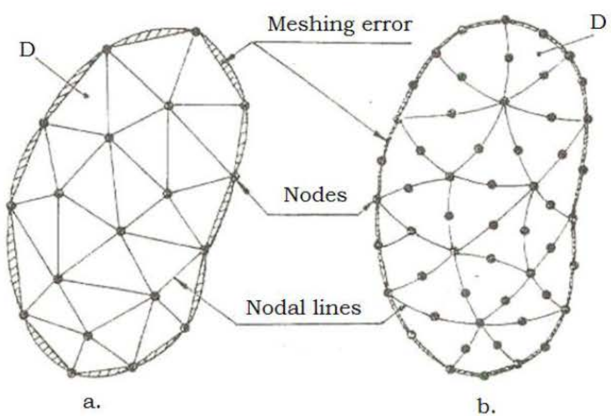


Fig. 75 Meshing elements [40]

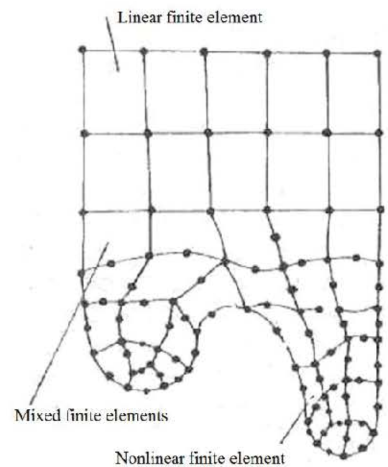


Fig. 76 Mixed finite elements [40]

Continuous development of advanced software that are based on FEM was achieved by diversifying the types of finite elements in association with various physical phenomena analyzed. Although FEM is a general method for solving differential or integro-differential equations of governing different physico-technical phenomena with various initial and boundary conditions, from reasons of productivity and economy, finite elements with an increased degree of generality have not been developed but were defined and modeled finite elements customized for different domains (1D, 2D, 3D), low levels of approximation (linear and parabolic algebraic functions), materials (linear, nonlinear) and types of problems (static, dynamic, and so on).

Generally, in terms of types of geometric domains to model, finite elements can be one-dimensional for the linear type of geometric domain, two-dimensional, for surfaces; three-dimensional, for volumes. These finite elements in terms of approximation function may be linear, straight edge lines, or non-linear, with curved edges. Typically, nonlinear finite elements by the type of polynomial approximation used, there may be quadratic parabolics (of second order), with one intermediate node, or cubic (third order) with two intermediate nodes.

3.3.4. Types of Finite Elements [17]

3.3.4.1. Element Dimensionality

Elements can have intrinsic dimensionality of one, two or three space dimensions. There are also special elements with zero dimensionality, such as lumped springs or point masses. The intrinsic dimensionality can be expanded as necessary by use of kinematic transformations. For example a 1D element such as a bar, spar or beam may be used to build a model in 2D or 3D space.

3.3.4.2. Element Nodes

Each element possesses a set of distinguishing points called nodal points or nodes for short. Nodes serve a dual purpose: definition of element geometry, and home for degrees of freedom. When a distinction is necessary we call the former geometric nodes and the latter connection nodes. For most elements studied here, geometric and connector nodes coalesce.

Nodes are usually located at the corners or end points of elements, as illustrated in (**Table 5**). In the so-called refined or higher-order elements nodes are also placed on sides or faces, as well as possibly the interior of the element.

From the point of view of geometrical or physics features required by analysis questions, the finite element modeled and implemented in a libraries of specific software applications, are different.

In some elements geometric and connection nodes may be at different locations. Some elements have purely geometric nodes, also called orientation nodes to complete the definition of certain geometric attributes.

In order to achieve meshing with various degrees of approximation of a geometric area, some of the FEM based software applications used mixed finite element in terms of the approximation (nodal lines straight and curves lines). Thus, Fig. 76 shows the plan area of the mesh of finite elements and non-linear; to achieve continuity of the structure of finite elements between the nodal areas discretized at the two types of finite elements is interposed an intermediate mixed discretized by finite elements having common sides of the same numbers of nodes.

3.3.4.3. Element Geometry

The geometry of the element is defined by the placement of the geometric nodal points. Most elements used in practice have fairly simple geometries. In one-dimension, elements are usually straight lines or curved segments. In two dimensions they are of triangular or quadrilateral shape. In three dimensions the most common shapes are tetrahedra, pentahedra (also called wedges or prisms), and hexahedra (also called cuboids or “bricks”).

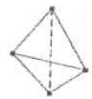
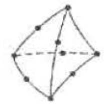
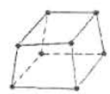
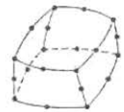
3.3.4.4. Element Degrees of Freedom

The element degrees of freedom (DOF) specify the state of the element. They also function as “handles” through which adjacent elements are connected. DOFs are defined as the values (and possibly derivatives) of a primary field variable at connector node points. Here we simply note that the key factor is the way in which the primary variable appears in the mathematical model. For mechanical elements, the primary variable is the displacement field and the DOF for many (but not all) elements are the displacement components at the nodes.

3.3.4.5. Nodal Forces

There is always a set of nodal forces in a one-to-one correspondence with degrees of freedom. In mechanical elements the correspondence is established through energy arguments.

Table 5 Main finite elements [40]

Type	Geometric shape	Approximation polynomial type		
		Linear	Quadratic	Cubic
1-D Element –line (Spring, truss, beam, pipe, etc.)	Line			
2-D Element - plane (Membrane, plate, shell, etc.)	Triangular			
	Quadrilateral			
3-D Element – solid (3-D fields - temperature, displacement, stress, flow velocity)	Tetrahedron			
	Pentahedron			
	Hexahedron			

3.3.4.6. Element Constitutive Properties

For a mechanical element these are relations that specify the material behavior. For example, in a linear elastic bar element it is sufficient to specify the elastic modulus E and the thermal coefficient of expansion α.

Table 6 Finite element types [40]

Geometry type	Specifics characteristics	Input parameters	Output parameters
1 D	Straight or curved bar subjected to traction-compression	Sectional parameter (area) The set of material parameters	Nodal displacements Elemental stresses Efforts Reaction forces
	Straight or curved bar subjected to traction-compression, shearing, torsion and bending	Sectional parameter (area and inertia moments) The set of material parameters	
2 D	Flat plate in a flat state of stresses	Thickness The set of material parameters	Nodal displacements Elemental stresses in different plans Nodal forces Reaction forces
	Flat plate in a flat state of strains	Thickness The set of material parameters	
	Spatial plate	Thickness The set of material parameters	
	Shell	Thickness The set of material parameters	
	Layered plate	Thickness The set of material parameters	
	Spatial structure in asymmetric stress state	The set of material parameters	Nodal displacements Elemental stresses Nodal forces Reaction forces
3 D	Spatial structure	The set of material parameters	

3.3.4.7. Element Fabrication Properties

For mechanical elements these are fabrication properties which have been integrated out from the element dimensionality. Examples are cross sectional properties of MoM elements such as bars, beams and shafts, as well as the thickness of a plate or shell element. For computer implementation the foregoing data sets are organized into data structures. These are used by element generation modules to compute element stiffness relations in the local system.

In (**Table 5**) illustrates the main finite elements contained by commercial advanced analysis programs with FEM, indicating the main input and output parameters. As a result, specific finite element are found, imposed by the type of geometric domain (bars, plates, membranes and massive spatial structures), the resistance of structures (bars resistant to traction-compression or traction-compression, shearing, torsion and bending), the stress and deformation states (surfaces in a plane state of stresses deformations or axisymmetric spatial structures), the type of input/ output parameters and the internal configuration of the material (massive, laminated structures) [40].

In addition to these types of finite elements that have a structural nature, most advanced programs also have specialized unstructural elements (Rigid, Spring, etc.) for modeling mechanical bond problems and increased degree of idealization modeling (mass concentrated rigid spring, spring-damper etc.), and for total or part modeling of structures, when accuracy and cost are appropriate. **Table 6** shows the main unstructural element groups, contained in the groups mentioned above, which are frequently encountered in commercial software for the analysis of the mechanical fields.

Table 7 Input parameters of finite elements

<table>
<tr><th>Type of finite element</th><th>Characteristics</th><th>Input parameters</th><th>Output parameters</th></tr>
<tr><td>Inertial, mass</td><td>Mononodal</td><td>Masses, inertial moments and inertial matrixes</td><td>Nodal displacements
Nodal forces</td></tr>
<tr><td>Rigid</td><td>Multinodal</td><td>Degrees of liberty with canceled relative displacements</td><td>Nodal displacements
Nodal forces</td></tr>
<tr><td>Arc</td><td>Binodal</td><td>Linear and torsional rigidity</td><td rowspan="2">Nodal displacements
Nodal forces
Internal forces</td></tr>
<tr><td>Damper-arc</td><td>Binodal</td><td>Rigidity and dampering constant</td></tr>
</table>

Inertial or mass elements shapes the structure of an element of a mechanical system or parts of it by reducing it to a material point to which it is assigned mass properties and/ or inertia equivalent properties.

Simplified modeling of mechanical structure areas with increased rigidity can be made with rigid finite elements that introduce one or more nodes with invariable relative positions as degrees of freedom defined above, to another node. In contrast to this possibility of modeling of mechanical system elements or parts of them can be replaced with elements that reduce their structure down to two material points (nodes) connected by an arc (arc finite element) or through an damper-arc system (finite element of a damper-arc type) which are inserted as input data accordin to the corresponding characteristics.

3.3.5. Classification of Mechanical Elements [17]

The following classification of finite elements in structural mechanics is loosely based on the "closeness" of the element with respect to the original physical structure. It is given here because it clarifies points that recur in subsequent sections, as well as providing insight into advanced modeling techniques such as hierarchical breakdown and global-local analysis.

3.3.5.1. Primitive Structural Elements

These resemble fabricated structural components. They are often drawn as such; see (Fig. 77). The qualifier primitive distinguishes them from macroelements, which is another element class described below. Primitive means that they are not decomposable into simpler elements.

These elements are usually derived from Mechanics-of-Materials simplified theories and are better understood from a physical, rather than mathematical, standpoint. Examples are: bars, cables, beams, shafts, spars.

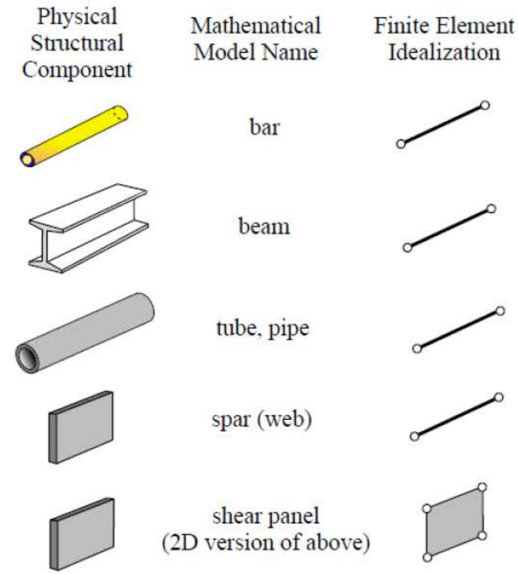


Fig. 77 Examples of primitive structural elements [17]

3.3.5.2. Continuum Elements

These do not resemble fabricated structural components at all. They result from the subdivision of "blobs" of continua, or of structural components viewed as continua.

Unlike structural elements, continuum elements are better understood in terms of their mathematical interpretation. Examples (Fig. 78): plates, slices, shells, axisymmetric solids, general solids.

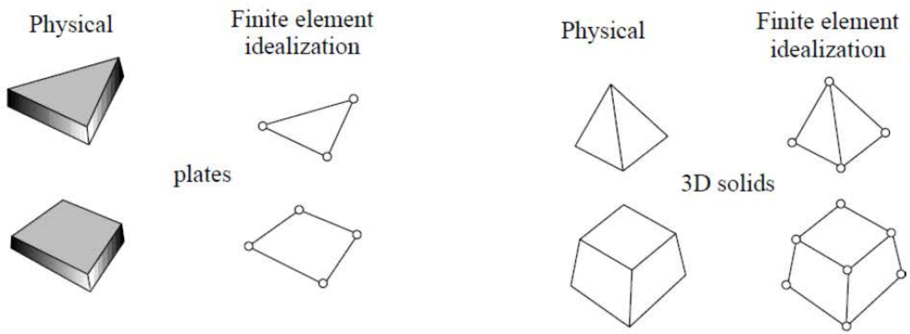


Fig. 78 Continuum element examples [17]

3.3.5.3. Special Elements

Special elements partake of the characteristics of structural and continuum elements. They are derived from a continuum mechanics standpoint but include features closely related to the physics of the problem. Examples: crack elements for fracture mechanics applications, shear panels, infinite and semi-infinite elements, contact and penalty elements, rigid-body elements (Fig. 79).

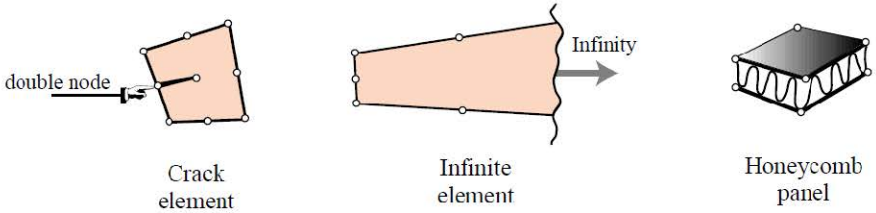


Fig. 79 Special element examples [17]

3.3.5.4. Macroelements

Macroelements are also called mesh units and superelements, although the latter term overlaps with substructures (defined below). These often resemble structural components, but are fabricated with simpler elements (Fig. 80).

The main reason for introducing macroelements is to simplify preprocessing tasks. For example, it may be simpler to define a regular 2D mesh using quadrilaterals rather than triangles. The fact that, behind the scene, the quadrilateral is actually a macroelement may not be important to most users.

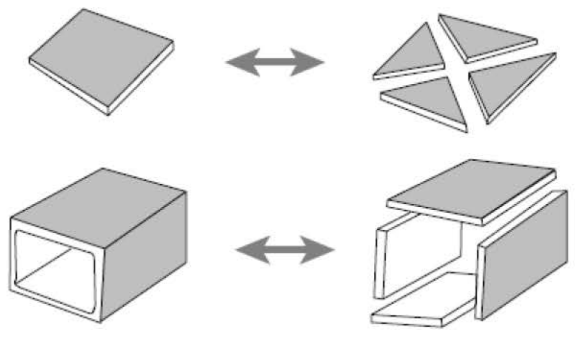


Fig. 80 Macroelements examples [17]

Similarly a box macroelement can save modeling times for structures that are built by such components; for example box-girder bridges.

3.3.5.5. Substructures

Also called structural modules and superelements. These are sets of elements with a well defined structural function, typically obtained by cutting the complete structure into functional components.

Examples: the wings and fuselage of an airplane; the towers, deck and cables of a suspension bridge. The distinction between substructures and macroelements is not clear-cut. The main conceptual distinction is that substructures are defined "top down" as parts of a complete structure, whereas macroelements are built "bottom up" from primitive elements. The term superelement is often used in a collective sense to embrace element groupings that range from macroelements to substructures.

3.3.6. Assembly [17]

The assembly procedure of the Direct Stiffness Method for a general finite element model follows rules identical in principle to those discussed for the truss example. As in that case the processs involves two basic steps:

- **I.** Globalization. The element equations are transformed to a common global coordinate system, if necessary.
- **II.** Merge. The element stiffness equations are merged into the master stiffness equations by appropriate indexing and matrix -entry addition.

The hand calculations for the example truss conceal, however, the implementation complexity. The master stiffness equations in practical applications may involve thousands or even millions of freedoms, and programming can become involved.

3.3.7. Modeling methods with elements [40]

The dividing operation (meshing) of the geometric domain of the problem to be analyzed into subdomains is complex, with implications both on the accuracy of results and the duration (cost) of the analysis.

Finite element modeling of the domain geometric of the analyzed problem using advanced programs that are based on FEM, can be achieved in one of the ways described below.

Generating the mesh directly without a geometric model, lately rarely used, is used for problems with very simple geometries, for modifying or supplementing existing finite element models and for obtaining unstructural elements.

Generating the messh indirectly starts from a geometric model or a previously generated finite element model. Most advanced programs have commands that allow meshing primary geometric entities such as lines, surfaces and volume. In the last years the advanced packs for finite element analysis have automatic meshing of complex geometrical domains. Automatic meshing, even with the mentioning of the ideal finite element dimension and/ or position of nodes on the domain boundary, thanks to the modeling performance of finite elements and geometric domain complexity sometimes leads to structures with inappropriate finite elements (with increased deviations from the ideal form and / or dimensional jumps.

In many cases, in order to obtain a finite element structure, there are commands which allow to copy a part of the finite element model. In Fig. 81 illustrates, for example, the process of copying through translation (Fig. 81, b) and, by a 180° rotation (Fig. 81, c) a one-dimensional finite element structure shown in Fig. 81, a.

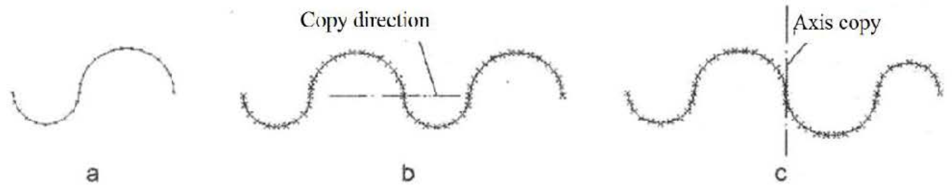


Fig. 81 Process of copying [40]

Useful for finite element modeling are also the generating controls of finite elements, two-dimensional or three-dimensional through extrusion of translation or rotation of structures with finite elements of inferior order, one-dimensional or two-dimensional.

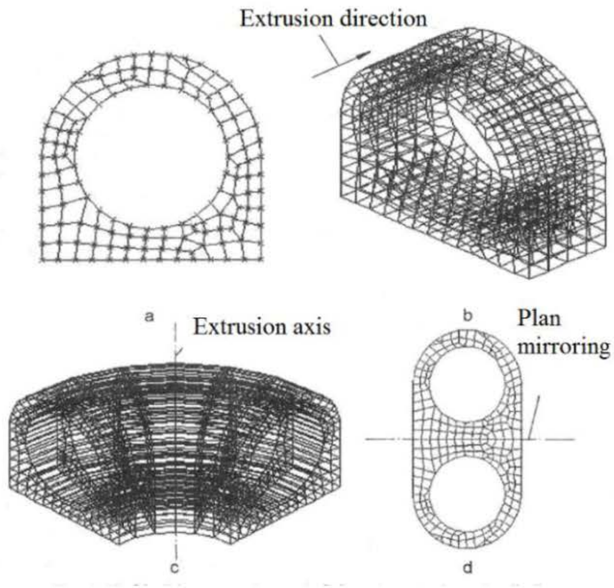


Fig. 82 Two- and three-dimensional FE structures [40]

For example, Fig. 82, b, and c illustrate the three-dimensional finite element structures generated by extruding the translation using a vector, respectively, by a 90° rotation around an axis,of the two-dimensional finite element structure shown in Fig. 82, a. In Fig. 82, d it can be seen the structure of two-dimensional finite element obtained by mirroring to a plane of a structure from Fig. 82, a.

3.3.8. Choosing finite elements and meshing parameters

The adoption of finite element types and the mesh sensitivity of geometric model of a problem is done taking into account the aspects of economic implications (time solving, methodologies of analysis and pre-and postprocessing possibilities) and the expected accuracy of the results.

For many practical problems, the geometrical shape of the domain to be analyzed, correlated with external loads, represents a primary source of information regarding the group of which the finite element in the process of being used is a part of. Thus, for example, in the cases of structures shown in Fig. 83 a, b and c, one-dimensional, two-dimensional, respectively, three-dimensional finite elements can be adopted,. But the optimal choice of finite elements is not always as simple as it may seem at the first analysis of the problem, choosing solely by intuition, in many cases, proved not to be suitable enough.

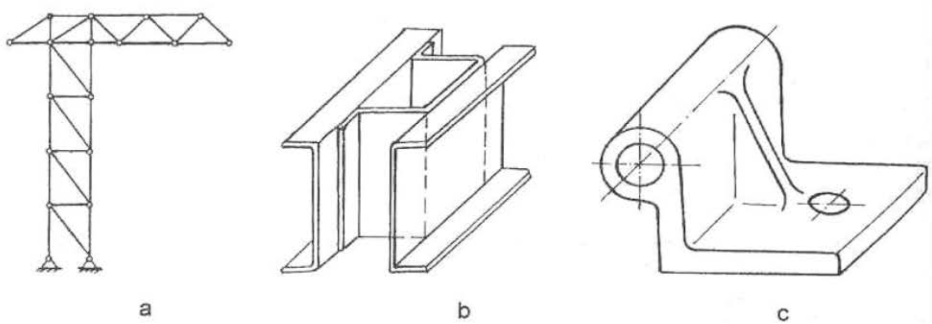


Fig. 83 One-, two- and three-dimensional structures [40]

So, adopting appropriate finite element types and mesh parameters (predefining finite elements sizes) to solve a given problem involves experience in this activity. It is acquired through testing and repeated confrontations of analysis results of the same problem using different types of finite elements and mesh. The efficiency of a finite element model analysis, customized by types of finite elements used in correlation with the sensitivity degree of the mesh, is cuantified through the accuracy of the results obtained and the time used to solve the associated numerical model.

Adopting the meshing parameters which predefine the size and number of finite elements is being produced so as to obtain a minimum difference between the approximate solution (obtained by finite element analysis) and the exact one (Fig. 84). The close result between the solution obtained and the exact one with increasing number of finite elements is called convergence.

Theoretically, at the limit, if the size of the finite elements become infinitely small, the exact solution would be obtained. Fig. 84 [40] allows to observe that there is a number of finite elements, N_e, which, if exceeded, does not lead to a significant increase in the convergence calculation in order to justify the additional calculating effort required.

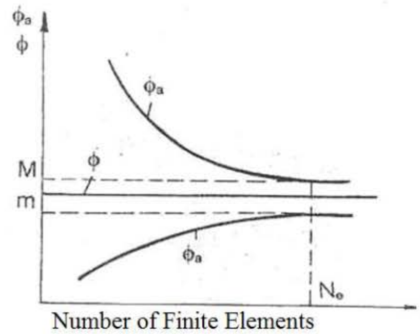


Fig. 84 The convergence to a theoretical solution [40]

Consequently, for the solution sought by the finite element analysis there is an superior bound, M, or an inferior one, m, which tends toward convergence process. In addition, it is also influenced by the type of approximation functions of the physical parameter, attached to a chosen finite element that, in the case of isoparametric finite elements, are the same with the approximation functions of the geometric domain. So, to improve convergence, structures of higher order finite elements can be achieved (approximation functions with high polynomial degree), presenting a lower degree of accuracy or lower degree meshing parameters with increased accuracy.

The geometric positioning of nodes in the analyzed problem, is done automatically in the process of meshing by algorithms that need to locate nodes also in areas where geometrical discontinuities (dimensional jumps) and physical (material inhomogeneities, loading concentration and / or irregularities) and imposed boundary conditions occur.

In many engineering problems, for the finite element modeling of the geometric domain, it is necessary to use two or more types of finite elements. Often encountered in practice are the situations in which one-dimensional finite elements and the two-dimensional ones engage with one another. For example, for shaping the wing structure shown in Fig. 85, bar-shaped finite elements for modeling ledges/strips and spar bases, and finite elements classified under plate-type objects (in a plane state stress) and shell-shaped objects for modeling ribs, spars and respectively the coating.

In case of concentration areas of unknown fields for achievement their maximum values, it is necessary to increase the mesh accuracy or the use of superior finite elements (with an increased degree of approximation) in these areas (Fig. 86).

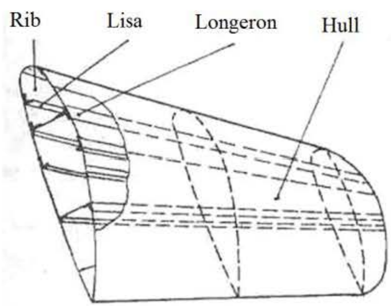


Fig. 85 Wing structure [40]

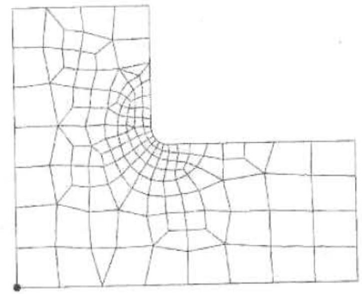


Fig. 86 Concentration area [40]

The accuracy of finite element analysis is influenced by the way the geometric approximation outlines how to analyze the domain problems. Triangular and tetrahedral finite elements provide broader possibilities for approximating the areas bounded by lines and, respectively, by surfaces compared to rectangular ones, respectively, tetrahedral which on the other hand reproduce more coherently stress distribution [40]. Improving the approximation of complex contours of geometrical domains can be done by increasing the number of linear finite elements (straight nodal lines) or by using nonlinear finite elements (with intermediate nodes on nodal lines), but fewer in number.

On the other hand, the finite element model convergence and accuracy are also influenced by the approximation of the finite element shape to the adopted associated regular shapes (equilateral triangle, square, regular tetrahedron and cube) considered as ideal finite elements.

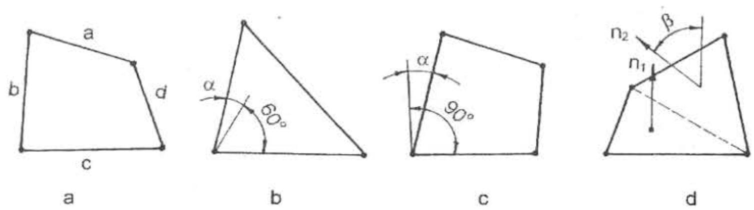


Fig. 87 Associated regular shapes [40]

Assessing the obtained finite element deviations as a result of meshing the ideal one, in many advanced programs based on FEM, for example, can also be made by means of the value parameters :

- the appearance factor, as ratio between the maximum and minimum edge length of the finite element (max {a, b , c , d } / min {a, b , c , d } , Fig. 87, a);
- the narrowing factor, as the maximum ratio of opposite sides of the finite element (max {a / c , c / a , b / d, d / a } , Fig. 87, a);
- the inner angles deviation as the maximum angle deviation of the finite element starting from the value of 60 ° or 90 ° of the ideal element (α , Fig. 87, b, c);
- the smoothness deviation, as the maximum of dihedral angles between normal planes grouped in sets of three nodes (β , Fig. 87, d).

Most advanced programs that are based on FEM allow to analyze the shapes of finite elements and indicate boundary values for the parameters quantifying their deviations from the ideal form.

Each type of finite element used to mesh the geometric model of the analyzed problem is associated with a set of specific parameter values that describe geometric (area, moments of inertia, thickness, etc.) and physical properties (mechanical and thermal characteristics of the material, density, damping coefficient, reference temperature and so on). To enter and modify specific parameter values, advanced software packages that are based on FEM have synthetic commands or menu systems appropriate to each group of finite elements.

3.3.9. Meshing example (1D, 2D, 3D)

In order to illustrate how the mesh of mechanical structures is made, we proposed following didactical example, which can be discretized using finite elements of three types:1D, 2D and 3D (Fig. 88).

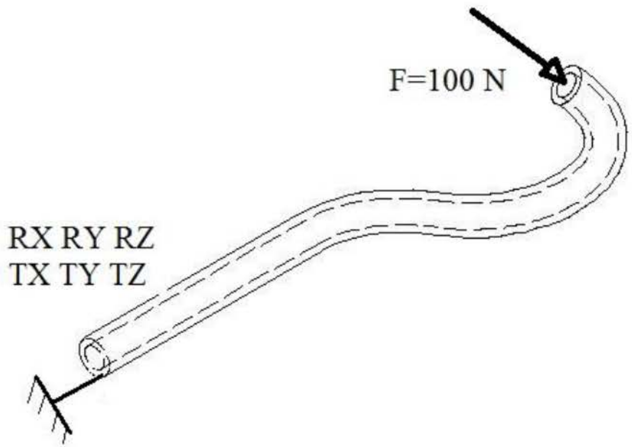
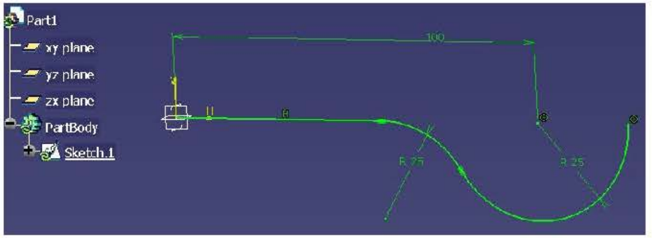
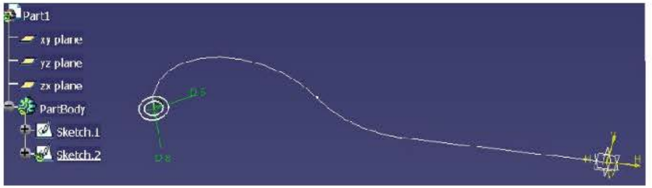
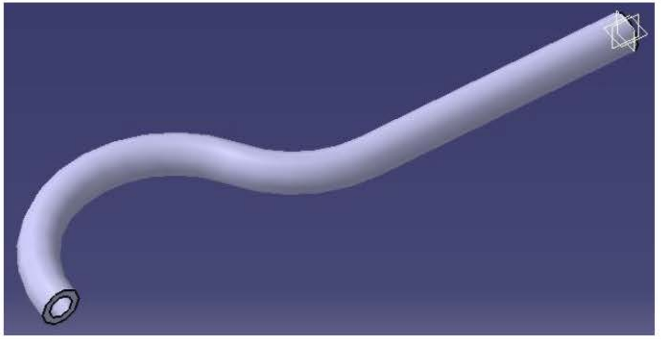


Fig. 88 Model for analysis

This example is made using CATIA software / Part Design module.

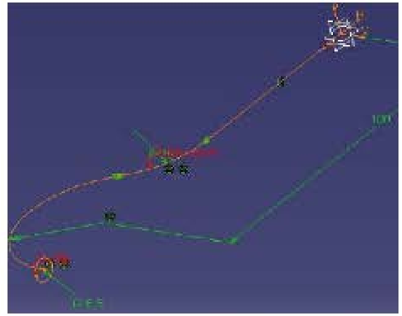
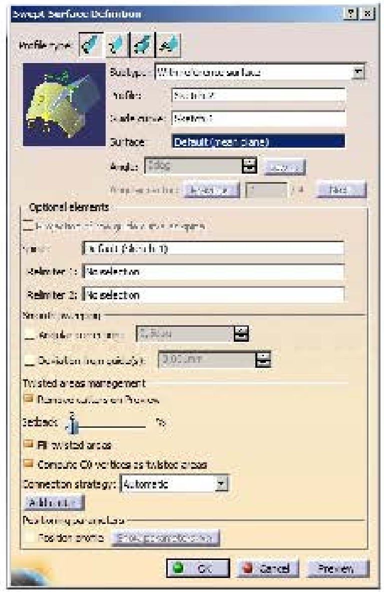
3.3.9.1. Meshing in CATIA with 3D finite elements

Step 1. Drawing the axis of the piece in *Part Design* module (here in *Sketch* module)	
Step 2. Drawing the profile (back in *Part Design*)	
Step 3. Extrude the beam profile along the axis (*Part Design*)	

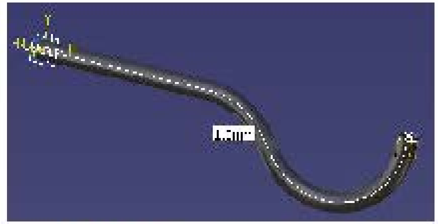
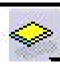
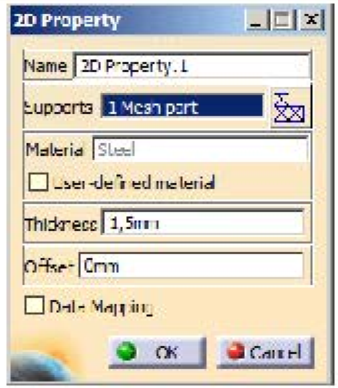
Step 4. Going in *Generative Structural Analysis* module and automaticaly mesh	
Step 5. Changing the meshing properties (size of the FEs and sag) by double clicking on the *Nodes and Elements.*	

3.3.9.2. Meshing in CATIA with 2D finite elements

The model is the same as in section 3.3.10.1.

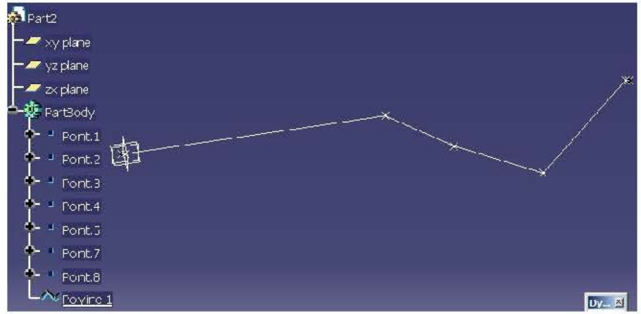
Step 1. Going in *Shape / Generative Shape Design* and draw the axis of the piece. **Step 2.** Drawing the circular profile of the piece, at the end of the axis	
Step 3. Using the *Swep* command , *Swept Surface Definition,* make the profile of the piece along the axis.	

Step 4. Saving profile and attaching the material.	
Step 5. Going to *Analysis & Simulation / Advanced Meshing Tools.*	
 **Step 6.** Using *Surface Mesher* command, you can change the properties of the meshing.	

Step 7. Going in *Analysis & Simulation / Generative Structural Analysis,*	
Step 8. Using  *2D Property* commands, you can change the properties (thickness) of the piece.	
The results of the meshing will be as shown in the figure.	

3.3.9.3. Meshing in CATIA with 1D finite elements

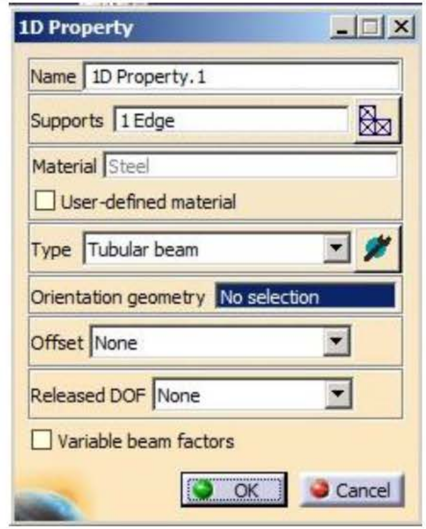
The model is the same as in section 3.3.10.1. and 3.3.10.2.

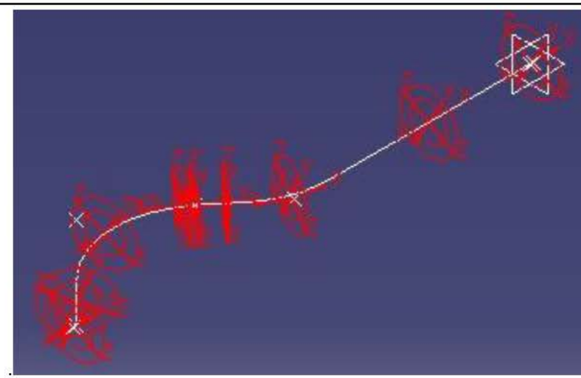
Step 1. Going in *Mechanical Design / Part Design* and draw the 8 points using *Point* command . **Step 2.** Use *Polyline* command , and *Polyline Definition* in order to define the radius of connections. Attaching a material.	
Step 3. *Analysis & Simulation /Advanced Meshing Tools:* New Analysis, Case: Static Analysis, OK.	

Step 4. Introduction of finite element type and meshing characteristics using *Beam Mesher* command .

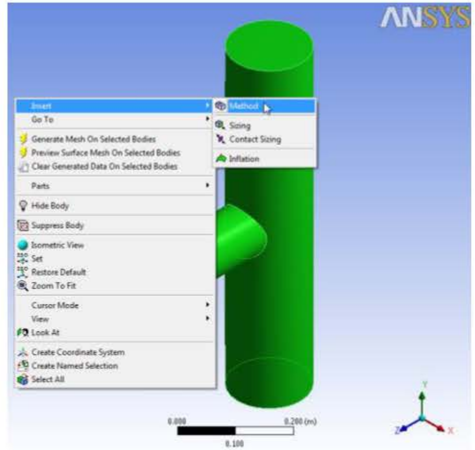
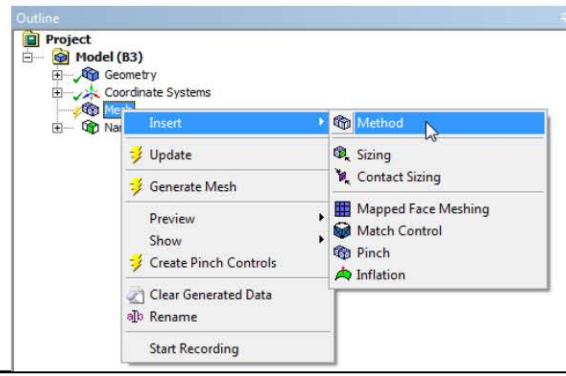
Step 5. Introducing finite element properties: *Analysis & Simulation / Generative Structural Analysis* by using

1D Property command and *Beam Definition* command .



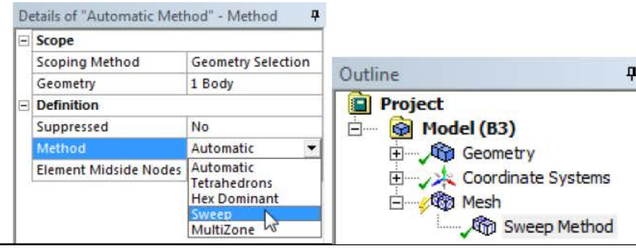


3.3.9.4. Meshing in ANSYS – Inserting Methods [6]

Ver. 1 - In the Outline, right click Mesh, *Insert > Method;* – Select body in Details View	
Ver.2 - In the Graphics Window, Select body(s) , right click, *Insert > Method;* – Body automatically selected in Details View	

Method is selectable using the drop down box

– Select, Automatic, Tetrahedrons, Hex Dominant, Sweep or Multizone (Automatic is used where no method has been explicitly specified)



3.3.9.5. Meshing in ANSYS – Automatic Methods [6]

Method Behavior

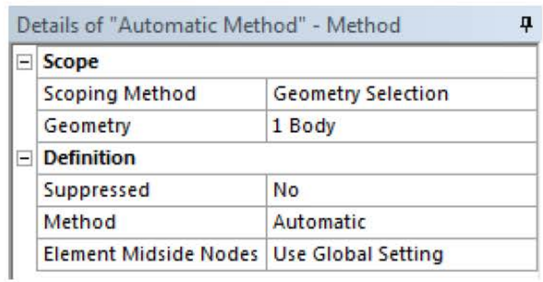
• Combination of Tetrahedron Patch Conforming and Sweep Method (Automatically identifies sweepable bodies and creates sweep mesh; All non - sweepable bodies meshed using tetrahedron method)

• Details of Tetrahedron Patch Conforming & Sweep

Access

• Default Method where not specified

• Can specify by inserting Method and setting to Automatic



3.3.9.6. Meshing in ANSYS – Tetrahedrons Methods [6]

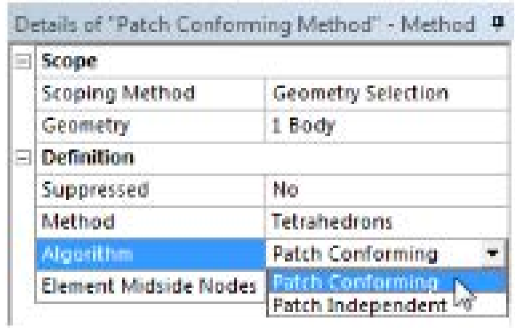
Method Behavior Generates tetrahedral elements - two algorithms are available:	✓ Patch Conforming ✓ Patch Independent
Patch Conforming	

Method & Algorithm Behavior

- Bottom up approach: Meshing process starts from edges, faces and then volume
- All faces and their boundaries are respected (conformed to) and meshed
- Good for high quality (clean) CAD geometries
- Sizing is defined by global and/or local controls

Access

- Insert Method and set to Tetrahedrons
– Additional drop down box for algorithm choice appears
- Set Patch Conforming
– Patch Conforming Method listed in Outline under Mesh object



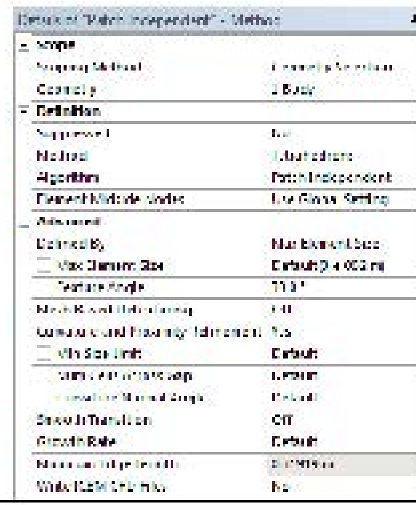
Patch Independent

Method & Algorithm Behavior

- Top down approach: Volume mesh generated first and projected on to faces and edges
- Faces, edges and vertices not necessarily conformed to
– Controlled by tolerance and scoping of Named Selection, load or other object
- Good for gross de - featuring of poor quality (dirty) CAD geometries
- Method Details contain sizing controls

Access

- Insert Method and set to Tetrahedrons
– Additional drop down box for algorithm choice appears
- Set Patch Independent



3.4. Boundary conditions. Introducing supports

3.4.1. Boundary Conditions [17]

A key strength of the FEM is the ease and elegance with which it handles arbitrary boundary and interface conditions. This power, however, has a down side. A big hurdle faced by FEM newcomers is the understanding and proper handling of boundary conditions. Below is a simple recipe for treating boundary conditions.

3.4.1.1. Essential and Natural B.C.

The key thing to remember is that *boundary conditions* (BCs) come in two basic flavors: essential and natural.

- *Essential BCs* directly affect DOFs, and are imposed on the left-hand side vector *u*.
- *Natural BCs* do not directly affect DOFs and are imposed on the right-hand side vector *f*.

The basic recipe is:

Q1. If a boundary condition involves one or more degrees of freedom in a direct way, it is essential. An example is a prescribed node displacement.

Q2. Otherwise it is natural.

The term "direct" is meant to exclude derivatives of the primary function, unless those derivatives also appear as degrees of freedom, such as rotations in beams and plates.

3.4.1.2. B.C. in Structural Problems

Essential boundary conditions in mechanical problems involve displacements (but not strain-type displacement derivatives). Support conditions for a building or bridge problem furnish a particularly simple example. But there are more general boundary conditions that occur in practice. Astructural engineer must be familiar with displacement B.C. of the following types.

- *Ground or support constraints.* Directly restraint the structure against rigid body motions. Symmetry conditions. To impose symmetry or antisymmetry restraints at certain points, lines or planes of structural symmetry. This allows the discretization to proceed only over part of the structure with a consequent savings in modeling effort and number of equations to be solved.
- *Ignorable freedoms.* To suppress displacements that are irrelevant to the problem. In classical dynamics these are called ignorable coordinates. Even experienced users of finite element programs are sometimes baffled by this kind. An example are rotational degrees of freedom normal to smooth shell surfaces.
- *Connection constraints.* To provide connectivity to adjoining structures or substructures, or to specify relations between degrees of freedom. Many conditions of this type can be subsumed under the label multipoint constraints or multifreedom constraints. These can be notoriously difficult to handle from a numerical standpoint.

3.4.2. Introducing suports

The finite element structure analysis is subject to a set of boundary conditions which, in the case of mechanical field problems, have to at least cancel possible kinematic displacements under the action of the loads placed.

The removal of kinematic movements of with the analyzing finite elements structure is done by the cancelling possible displacements (usually shifts) associated during preprocessing with geometric entities and, after meshing, with nodes in accordance with the coordinate system axes (global or local), preliminarily adopted.

Choosing the geometric entities, nodes and degrees of freedom with canceled displacements is done while modeling so that after the analysis it results an increased closeness to the real model. Also, in the case of boundary conditions modeling, advanced programs can perform analyses with several sets (variants) of boundary conditions, so by comparison, the most unfavorable state of structure strain can be identified.

The visualization using the analysis model of canceled displacements associated to the points (nodes) in relation to the straight or circular axes of the coordinate system used, can be made by combining the appropriate symbols corresponding to rotation and translation displacement in **Table 3**. These are used for visualizing links to the point (node) level in relation to a tri-ortogonal coordinate system with the designated axes (straight line or circular) 1, 2 and 3.

Table 4. Reactions and displacements in joints [40]

In addition, shows the forces of reaction and the corresponding displacements, each associated to a symbol.

The boundary conditions ensure complete fixing of the base structure with respect to the base of the settlement in order to preventing the rigid body movements of the ensemble structure. In general, the boundary conditions imposed certain values of degrees of freedom in the restraint points of the structure. If the structure does not have enough circumstances bearing a certain load case can generate shifts with infinite value.

3.4.2.1. The minimum support for a plane structure (2D)

Regardless of the distribution of external forces, the boundary conditions must cancel the shift in relation to the two axes in two plane structure (x, y) and rotation about the axis normal to the plane of the structure (the axis z) - Fig. 89.

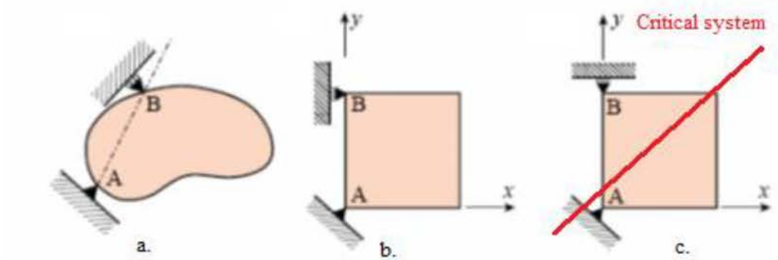


Fig. 89 Minimal conditions for restraints [17]

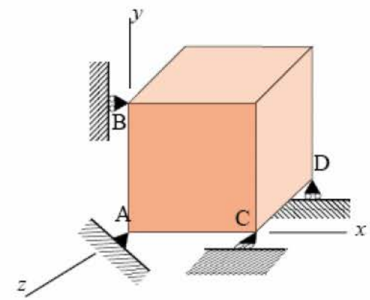


Fig. 90 Constraint of 3D structures [17]

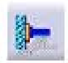
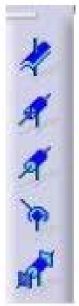
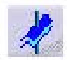
3.4.2.2. The minimum support for a volume structure (3D)

Regardless of the distribution of external forces, the minimum number is six and the boundary conditions have to cancel the three shift in relation to the three axes (x, y, z) and three rotations in relation to the axes correspond to those.

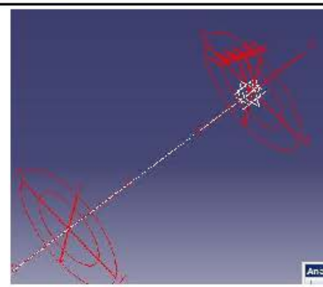
To fix the structure there are available numerous combinations of supports that ensures that the structure rigid body movements are prevented. A possible example, see the figure below (Fig. 90), where we used a hinged spherical bearing which block in point A the 3 possible translations of the body and three simple supports prevent rotation relative with: the axis (z) - bearing B, axis (y) - bearing C and axis (x) - bearing D.

3.4.3. Restraints in CATIA

The following table shows the controls modeling of various geometric constraints provided by the CATIA environment.

<table>
<tr><td rowspan="9">
</td><td colspan="2"></td><td>Clamp</td></tr>
<tr><td rowspan="5"></td><td></td><td>Surface slider</td></tr>
<tr><td></td><td>Slider</td></tr>
<tr><td></td><td>Sliding pivot</td></tr>
<tr><td></td><td>Ball joint</td></tr>
<tr><td></td><td>Pivot</td></tr>
<tr><td rowspan="2"></td><td></td><td>User – defined restraint</td></tr>
<tr><td></td><td>Isostatic restraint</td></tr>
</table>

Examples

<table>
<tr><td>Clamp on 3D finite elements</td><td>
</td></tr>
<tr><td>Clamp on 2D finite elements using a Rigid Virtual Part.</td><td>
</td></tr>
<tr><td>Clamp on 1D finite elements</td><td></td></tr>
</table>

3.4.4. Restraints in ANSYS

3.4.4.1. The joint feature [6]

The joint feature in Mechanical can provide a fast and simple alternative to contact when simulating the interaction between bodies or constraints to ground:

- While used extensively in rigid body analysis joints are not limited by body type and can be used in flexible and mixed rigid/flex models.
- Joints are defined in terms of their degrees of freedom with respect to a specific coordinate system (e.g. translation in the X direction or rotation about the Z axis).
- Joints are attached to bodies by scoping to a specific region of the part, a surface for example, just like contact.
- Contact pairs are defined as “contact” and “target” while joints use the terms “reference” and “mobile” to describe each “side” of a joint (for body to ground joints the ground is assumed to be the reference).

In the example shown here (Fig. 91), a body to ground revolute joint is scoped to a cylinder:

- The legend shows the “RZ” or rotation about Z is free.
- Degrees of freedom shown in grey are constrained.
- The “Reference Coordinate System” listed in the details is shown at the origin of the joint. This is the joint’s line of action.

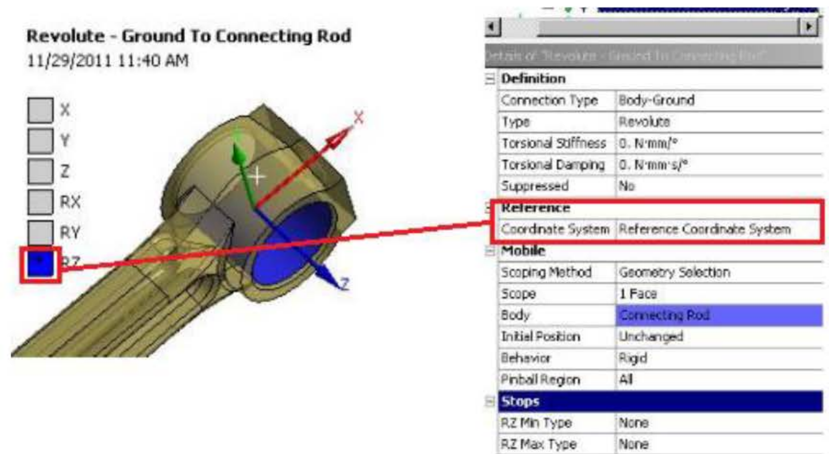


Fig. 91 Rotation joint in ANSYS [6]

3.4.4.2. Joint definitions [6]

There are 9 joint types available in Mechanical which can be either body to body or body to ground. In the revolute joint example below notice the reference and mobile regions are color coded.

The legend displays the joints behavior with respect to the reference coordinate system. Colored DOF are free, grey indicates a fixed DOF (Fig. 92).

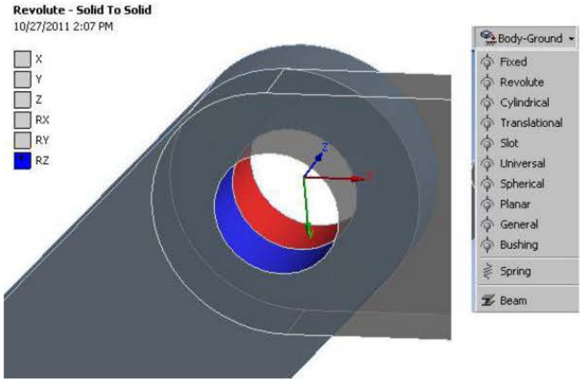


Fig. 92 Joint definitions in ANSYS [6]

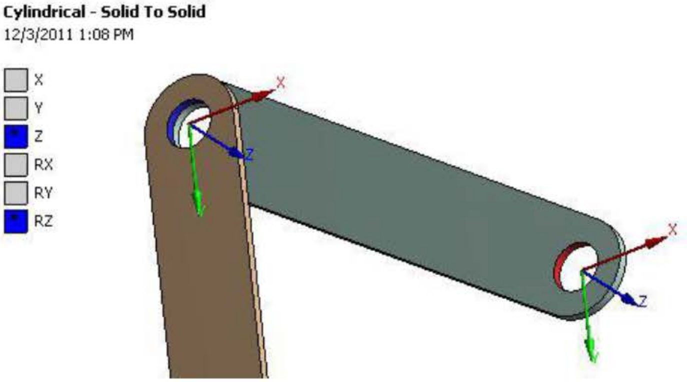


Fig. 93 Joint Coordinate Systems in ANSYS [6]

3.4.4.3. Joint Coordinate Systems [6]

All joints are defined in terms of 2 coordinate systems, the reference and mobile CS. The CS are associated with each part scoped to the joint. It's the relationship between the CS that controls the joints motions (Fig. 93).

3.4.4.4. Joint Configuration [6]

Configuring a joint allows the initial relationship between the reference and mobile coordinate systems to be changed:

• Begin by highlighting the joint to be configured in the tree.

• Now click the "Configure" icon in the context menu.

• When a joint is in configure mode its position can be changed by dragging the DOF handle shown below (Fig. 94).

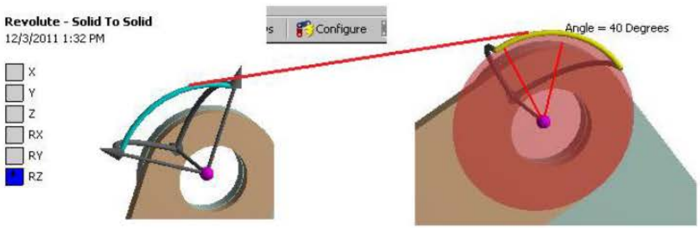


Fig. 94 Joint configuration [6]

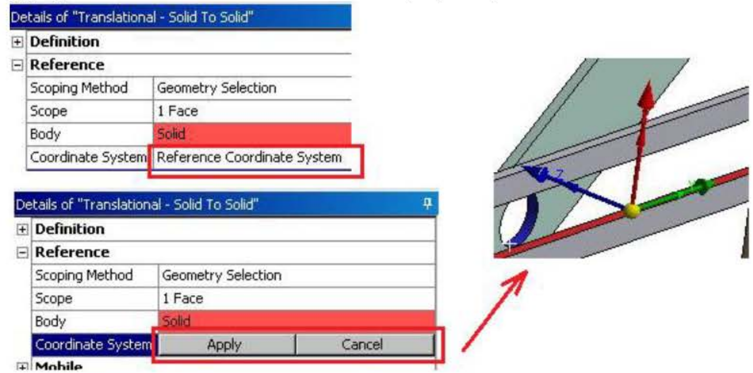
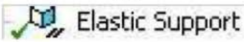
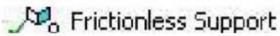
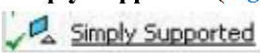
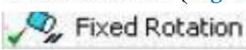


Fig. 95 Modifying joint coordinate system [6]

3.4.4.5. Modifying Joint Coordinate Systems [6]

Recall that a joint's motions are determined according the joint's coordinate systems. In some instances it will be necessary to reorient these systems to obtain the correct joint behavior. Click in the Coordinate System field in the details to bring up the apply/cancel buttons and place the coordinate system in edit mode. Notice the CS graphically expands while editing.

3.4.4.6. Supports in ANSYS [6]

Fixed Support 	Constraints all degrees of freedom on vertex, edge, or surface: – Solid bodies: constrains x, y, and z. – Surface and line bodies: constrains x, y, z, rotx, roty and rotz.
Displacement 	Applies known displacement on vertex, edge, or surface. • Allows for imposed translational displacement in x, y, and z (in user - defined Coordinate System). • Entering "0" means that the direction is constrained, leaving the direction blank means the direction is free.
Elastic Support 	Applies "flexible" frictionless support. • Foundation stiffness is the pressure required to produce unit normal deflection of the foundation.
Frictionless Support (Fig. 96) 	• Applies constraints (fixes) in normal direction on surfaces. • For solid bodies, this support can be used to apply a 'symmetry' boundary condition.
Cylindrical Support (Fig. 97) 	– Provides individual control for axial, radial, or tangential constraints. – Applied on cylindrical surfaces.
Compression Only Support (**Fig. 98**) 	• Applies a constraint in the normal compressive direction only. • Can be used on a cylindrical surface to model a pin, bolt, etc.. • Requires an iterative (nonlinear) solution.
Simply Supported (**Fig. 99**) 	• Can be applied on edge or vertex of surface or line bodies • Prevents all translations but all rotations are free
Fixed Rotation (**Fig. 100**) 	• Can be applied on surface, edge, or vertex of surface or line bodies • Constrains rotations but translations are free

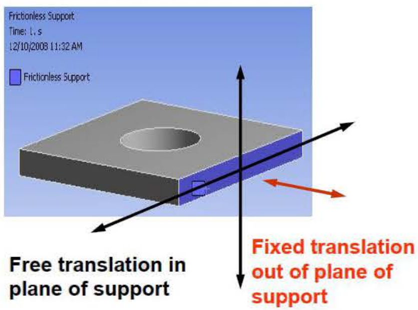


Fig. 96 Frictionless Support [6]

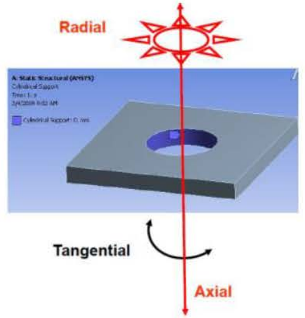


Fig. 97 Cylindrical Support [6]

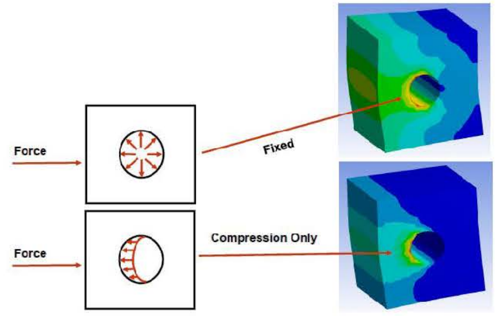


Fig. 98 Compression Only Support [6]

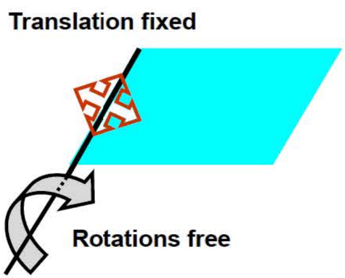


Fig. 99 Simply Supported Edge [6]

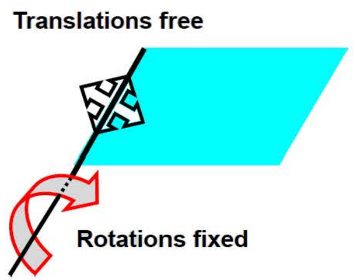


Fig. 100 Fixed Rotation Edge [6]

3.5. Load modeling

Finite element modeling and analysis of mechanical structures consider loads as initial, known data that are introduced in various ways so as to achieve a low rate of errors. Most advanced programs that are based on FEM allow the model analysis with the consideration of multiple sets (variants) of possible loads, in order to highlight the most unfavorable states of strain.

The loads acting on a finite element modeled mechanical structure, usually in the form of forces (moments), movements and temperatures, variation in space can be concentrated or distributed, using the variation in time - static or dynamic.

Concentrated loads are considered to be acting on the finite element structure nodes. They are actually in theory quasi-singularities, next to them resulting cvasiinfinite stresses. However, in computer-aided design, loads can be used as concentrated forces when studying displacement fields, strains and stresses in remote areas from their application nodes.

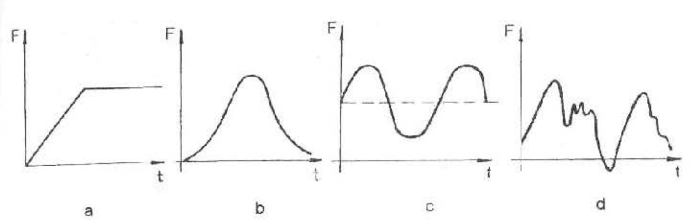


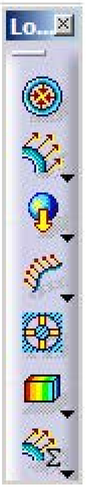
Fig. 101 Different trends of loads [40]

Distributed loads, with different laws of variation (constant linear, parabolic, etc.) may interact externally - on a line or on a surface - or internally – in volume. Using these loads allows modeling with reduced deviations from the real case, and thus, unlike the case of using concentration tasks, the fields from the loading areas can be considered as reliable for designing. The possibility of taking into consideration the distributed forces of inertial type (linear, centrifugal and/ or mass) also leads to the increase in design accuracy and, therefore, in the results obtained.

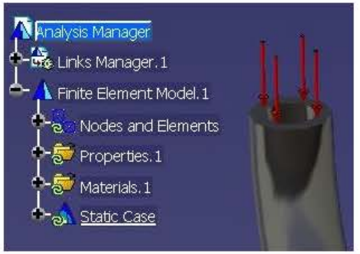
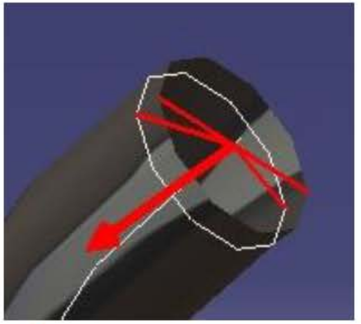
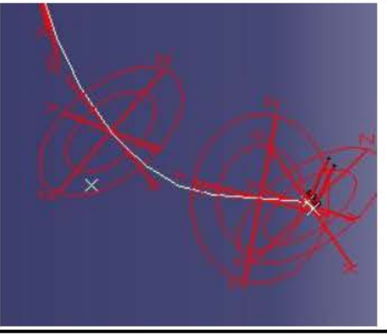
Loading is considered as being static when its value slowly increases from zero to the nominal value (Fig. 101, a). The increase is so slow that inertial forces have extremely low values and, therefore, they can be neglected for the finite element analysis. Dynamic loads, variable in time, can present shock when large variations of intensity in short periods of time (Fig. 101, b), regular (Fig. 101, c) or, generally, random (Fig. 101, d) occur. In some practical situations when the load values pertaining to the type of force cannot be identified, but the displacement fields as a result of pretentioning montage or strain restriction are partially or entirely known, as input data for the analysis of movements can specify the structure nodes finite elements.

3.5.1. Loads in CATIA

The following table presents modeling commands load type forces, moments, temperature and movement provided by the CATIA environment.

	Pressure		Temperature Field
	Distributed Force		Temperature Field for Thermal Solution
	Moment		Combined Loads
	Bearing Load		Assembled Loads
	Imported Force		
	Imported moment		
	Acceleration		
	Rotation Force		
	Line Force Density		
	Surface Force Density		
	Volume Force Density		
	Force Density		
	Enforced Displacements		

Examples

Distributed forces on 3D finite elements	
Distributed forces on 2D finite elements using a Rigid Virtual Part.	
Distributed forces on 1D finite elements	

3.5.2. Loads in ANSYS [6]

Loads and supports are thought of in terms of the degrees of freedom (DOF) available for the elements used. In solids the DOF are x, y and z translations (for shells and beams we add rotational DOF rotx, roty and rotz). Supports, regardless of actual names, are always defined in terms of these DOF.

Boundary conditions can be scoped to geometry items or to nodes (depending on load type). Boundary conditions applied directly to nodes are covered in the second part of this section.

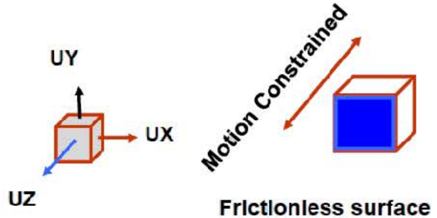


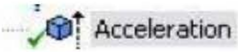
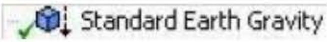
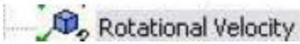
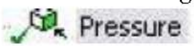
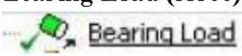
Fig. 102 Frictionless surface [6]

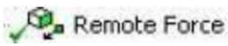
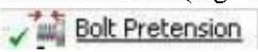
Load	Supports Coordinate Systems
Acceleration	No
Standard Earth Gravity	Yes
Rotational Velocity	Yes
Force	Yes
Remote Force	Location of Origin Only
Bearing Load	Yes
Moment	Yes
Given Displacement	Yes
Nodal Force	Yes

Fig. 103 Loads types in ANSYS [6]

Example: a "Frictionless Support" applied to the face of the block (Fig. 102) shown would indicate that the Z degree of freedom is no longer free (all other DOF are free).

Loads and supports having a direction component can be defined in global or local coordinate systems (Fig. 103):

Acceleration  	• Acts on entire model in length/time^2 units. • Acceleration can be defined by Components or Vector. • Forces resulting from accelerations act in the opposite direction of the acceleration (e.g. like being pushed back in the seat when an automobile accelerates forward).
Standard Earth Gravity  	• Choose the direction in which the force of gravity acts. • Value automatically set to current unit system. • Gravity direction can be defined in global or local coordinate systems.
Rotational velocity  	• Entire model rotates about an axis at a given rate. • Define by vector or component method. • Input can be in radians per second (default) or RPM.
Pressure loading  	• Applied to nodal groups or surfaces, acts normal to the surface. • Positive value into surface, negative value acts out of surface. • Units of pressure are in force per area.
Force loading (Fig. 104)  	• Forces can be applied on nodes, vertices, edges, or surfaces. • The force will be evenly distributed on all entities. Units are mass*length/time2. • Force can be defined via vector or component methods.
Hydrostatic Pressure (Fig. 105)  	• Applies a linearly varying load to a surface (solid or shell) to simulate fluid force acting on the structure. • Fluid may be Internal (contained fluid) or external (submerged body). User specifies: • Magnitude and direction of acceleration. • Fluid Density. • Free surface location of the fluid. • For Shells, a Top/Bottom face option is provided.
Bearing Load (force) (Fig. 106)  	• Forces are distributed in compression over the • projected area: ○ No axial components.

	○ Use only one bearing load per cylindrical surface. • If the cylindrical surface is split, select both halves of cylinder when applying the load. • Bearing loads can be defined via vector or component method.
Moment Loading (Fig. 107)  	• A moment can be applied to a vertex, edge, surface or nodes (named selection). • If multiple entities are selected, the moment load is evenly distributed. • Vector or component method can be employed using the right hand rule. • Units of moment are in Force*length.
Remote Force Loading (Fig. 108)  	• Applies an offset force on a vertex, edge, surface or nodes. • The user supplies the origin of the force (geometry or coordinates). • Can be defined using vector or component method. • Applies an equivalent force and moment on the surface
Bolt Pretension (Fig. 109)  	• Applies a pretension load to a solid cylindrical section or beam using: ○ Pretension load (force) – OR ○ Adjustment (length) • For body loading a local coordinate system is required (preload in z direction). • For sequenced loading additional options are available (see the Intro 2 course).
Thermal Loads (Fig. 110, Fig. 111) $\varepsilon_{th}^{x} = \varepsilon_{th}^{y} = \varepsilon_{th}^{z} = \alpha\left(T - T_{ref}\right)$	• Applies a uniform temperature in a structural • analysis. • Appears under "Loads" in structural analysis. • A reference temperature must be provided (can • apply to all bodies or individuals).

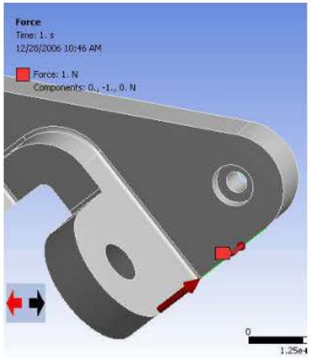


Fig. 104 Force loading [6]

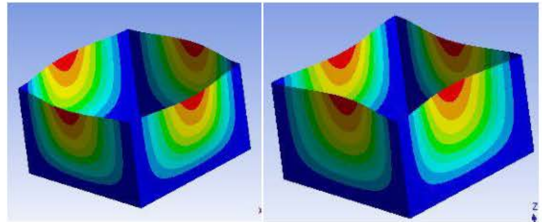


Fig. 105 Internal and external pressure [6]

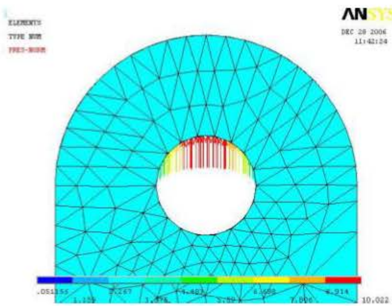


Fig. 106 Bearing load [6]

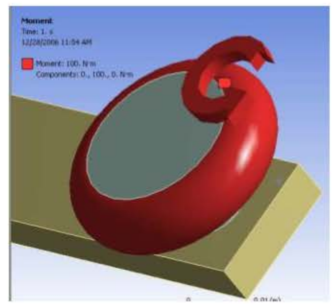


Fig. 107 Moment [6]

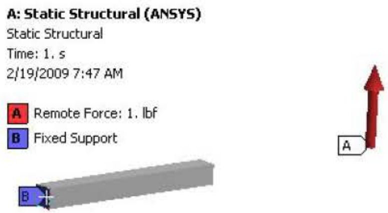


Fig. 108 Remote force [6]

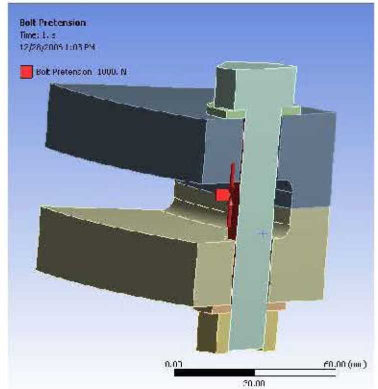


Fig. 109 Bolt pretension [6]

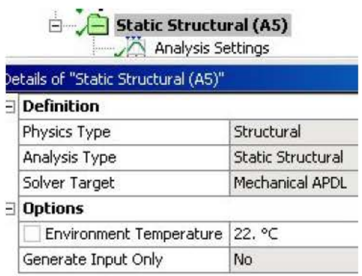


Fig. 110 Reference temperature in Environment

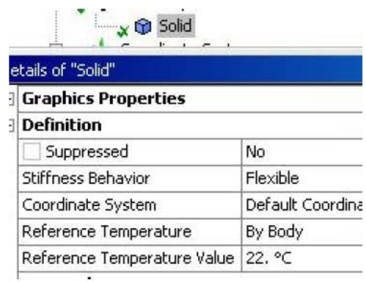


Fig. 111 Reference temperature can be applied to