1. CAD systems

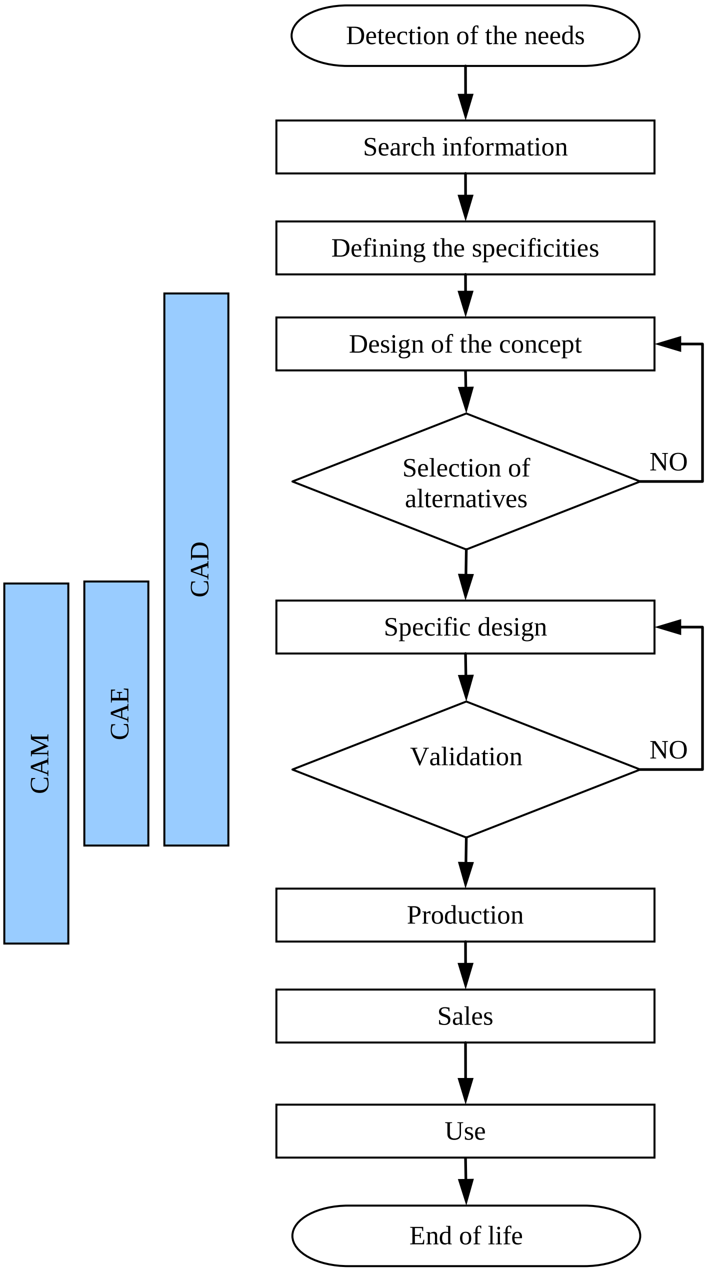
A Computer Aided Design (CAD) system integrates a machine or computer and an application program system.

The main characteristic of a CAD system is that it enables an interactive design.

CAD systems are combined with Computer Aided Engineering (CAE) and Computer Aided manufacturing (CAM) systems; therefore they become an important part of the automation of industrial processes. For a better explanation of this idea, the design process is illustrated in Image 1.1, where computer aided techniques are included.

Now a day, almost all the CAD offer is based on parametric and varying design. In a parametric design, all the parameters are controlled by the computer and assigned to specific features of the model. In addition, parametric design also enables the establishment of relationships between parameters.

A varying design permits the control of the parameters of the model along the different phases of the design process: part, assembly or technical drawing. As a result, all the information required for the design process is linked and therefore, every time a parameter is changed it is also automatically changed in the data base.



1.1. Image: Techniques based on CAD

1.1. Techniques for parametric modelling

Parametric modelling is the method used to design 2 or 3 dimensional models where the parameters can be modified. This characteristic that permits to change a model is possible due to the parameters defined when designing the model. In fact, these parameters define the model itself and its dimensions and geometry.

Parametric modelling enables the creation and variation of objects efficiently. If we compare this methodology with the traditional CAD/CAM modelling systems, the parametric modelling registers the rout and steps followed for the creation of the model, as well as the geometric relationships among all the entities. These relationships are saved in a file called parametric history.

The history saves all the operations that have been executed, including the order in which they were executed. This file can also be edited in order to vary the relationships and geometries defined before.

In this regard, the parametric history permits the modifications of the model and, as consequence, its reconstruction. If a parameter is changed, then all the geometries related to this parameter are also automatically modified. So, the model always maintains the geometrical relationships defined during the design.

In short, the parametric history automatically saves the following information:

- The values of the parameters, for instance, length, radius, diameter, etc. The values of the parameters can be modified by maintaining fixed geometrical relationships.
- Geometrical relationships such as relative positions or tangential positions. All type of geometries can be related one with another, as well as their reference systems (technical drawings).

Each of the steps performed in the parametric modelling will be related with a previously defined one or with an already existing geometry. In general, each step is related to an existing reference system or geometry, then, a new geometry is added. Thereby, the existing reference system and the geometry are always linked to the added new geometry.

The functional system of parametric modelling

Parametric modelling can be used for any type of representation, wires, surfaces or solids. The operation mode is the one explained bellow:

- Profiling: 2D interface can be used to generate free shape profiles. Then, the volume is added to the profiles.
- Creation of sketches: sketches are used to generate surfaces or solids from profiles. The profile can be generated in the 2D interface or from a previously created geometry. The relationship between the profile and the sketch is saved in the parametric history.
- Restrictions: this characteristic enables the definition of relationships by means of algebraic equations. For instance, it is possible to define that one of the surfaces of an object should have the same dimensions of another surface of another object. If one of the dimensions of both surfaces is modified, the other will also be modified automatically to fulfil the restriction requirement.
- Modelling of the basic elements: parametric modelling allows the creation of intelligent geometries and the addition of elements such as holes, chamfers, etc.

1.2. Modelling strategies

On the one hand, using the traditional modelling system (non-parametric) an object can be generated without having taken into account the procedure followed to obtain it. In the traditional modelling the aim of the design is not present. For instance, in the design of a plate containing a hole, either the plate or the hole can be generated in first place. Using this method, if an element is not correctly placed it can be selected and moved. Elements can be placed independently without connection with any other element.

On the other hand, in a parametric modelling, relationships are defined as the model is constructed. Hence, defining the relationships is the most important task.

Analysis of the model

In a parametric modelling the steps followed to generate the model are as important as the physical aspects of the model.

In first place, we have to examine the model and what we want to do with it, its aim, and consider how would the dimensions and the characteristics of the model vary. In fact, the parameterizations have to be flexible and capable to face all the possible variations.

The basic procedure to generate a parametric model can be shortly defined as:

1. Work on a plane.
2. Create the profiles.
3. Generate the solid.
4. Fulfil the requirements of the solid: chamfers, boolean operations and so on.

For each area of the model the steps are repeated and the operations are sequentially saved as a history file. The result is an object that can be assembled with other objects.

Parallel approach of the model

The parallel approach for modelling is more practical than the linear one. The parallel approach consists in generating each area of the model independently.

Using this method, the model is separated into functional blocks. Each block is generated independently without making any reference to other geometries of other functional blocks. Therefore, if the geometry of one of the functional blocks is modified, the rest of the blocks stay unaltered.

The following techniques are useful in a parallel approach of the model:

- The commands to link functional blocks are performed in the last step. In general, several operations such as boolean functions cannot be carried out independently, hence it is advisable to execute them at the end.
- If there exists a need to define restrictions among blocks, they must be defined by means of variables and equations.
- The modelling must be initiated from a geometric construction that will be the reference for the rest of the geometries.
- If a reference to an existing geometry is required, use an intermediate step: generate the geometry on an already existing geometry and reference it.

- If a reference to the end point of an existing axis is required, a dot must be placed in such point. If the axis is removed, then all the references are also removed. Placing a point we can avoid it.
- If a plane defined by the user is selected, before continuing with the modelling, we have to make sure the plane selected is one of the planes present in the system. The construction is registered by an active plane command. When a working plane is created, if the active plane command is deleted, then the system automatically will delete the plane that contains the construction active plane.
- Parameters can be defined in a single command in order to convert them into independent variables. For instance, instead of using “Insert Fillet Rad 5 # #” to generate a circle, “Insert Fillet Rad 5 # Rad 5 #” can be used. Same values can be assigned to the roundings using variables and equations and then, separate them. This enables modifications in the future.
- Free or independent positions are not modified along with the model, so they should be avoided. It is preferable to make references to geometrical constructions.
- Use the axes of the solid. As mentioned above, is preferable to make references to geometrical constructions. In addition, it is recommendable to provide two positions to define the vector direction.
- If a command generates several entities which are not of great interest, instead of erasing them, they should better be placed in a separate layer.
- Use the design command instead of deleting any entity.

Least dependence method

Least dependence method is a type of parametric modelling that breaks its basic laws.

There are several ways of creating a parametric model. "Least dependency" is one of the methods used to describe the model. This method is used by world business companies such as Rolls Royce Aerospace group to generate their parametric models.

This methodology consists in obtaining parametric entities with the least dependencies with other entities. By limiting the dependencies, future modifications in the model will not affect the rest of the entities.

This tactic leads to a logical, strong and flexible structure. A parametrically guided solid model is created with automatic updates among associated entities.

The benefits of the least dependency method are the following:

- Creation of strong models.
- Easy modifications of the geometry during the design process.
- Support to concurrent engineering.
- Time and cost saving while improving the quality.
- Simple and functional learning process.

The least dependency method offers a logical point of view of the parametric modelling technique. In this regard, a third person that did not create the model would easily understand it. In addition, less time is expected to execute the "replay history" command in order to understand the creation of the process.

1.3. Bibliography related to CAD Systems

BIANCONI, F.; CONTI, P.; DI ANGELO, L. *Interoperability among CAD/CAM/CAE systems: a review of current research trends*. Geometric Modeling & Imaging, New Trends; M. Sarfraz & E. Banissi editors, IEEE, 2006.

LÓPEZ SOTO, J. *Metodología de optimización del modelizado digital en el diseño paramétrico industrial*. Tesis doctoral. E.T.S. Ingenieros Industriales y Telecomunicaciones de Bilbao, Universidad del País Vasco / Euskal Herriko Unibertsitatea. 2010

LÓPEZ SOTO, J.; RAMÍREZ LÓPEZ-PARA, P.; CARO RODRÍGUEZ, J. L. *Aplicación del modelado paramétrico al diseño industrial*. Actas del XVI Congreso Internacional de Ingeniería Gráfica. Zaragoza, 2004. Pp. 413-422. I.S.B.N.: 84-95475-39-1

LÓPEZ SOTO, J.; RAMÍREZ LÓPEZ-PARA, P.; CARO RODRÍGUEZ, J. L. *El diseño de ingeniería desde el punto de vista paramétrico*. Actas del VIII Congreso Internacional de Ingeniería de Proyectos. Publicaciones de la Escuela Superior de Ingenieros de Bilbao, 2004. I.S.B.N.: 84-95809-22-2.

OLIVEIRA MATEUS, J. V.; HERNANDIS ORTUÑO, B. *Tecnologías CAD en design. La interfaz gráfica y el aprendizaje*. Actas del 20 Congreso Internacional de Ingeniería Gráfica. Universidad Politécnica de Valencia, 2008. Pp. 131-141. I.S.B.N.: 978-84-8363-275-8.