CAD BOOK

ÚJ SZÉCHENYI TERV

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A projekt keretében elkészült tananyagok:

Anyagtechnológiák

Materials technology

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Áramlástechnikai gépek

CAD tankönyv

CAD book

CAD/CAM/CAE elektronikus példatár

CAM tankönyv

Méréstechnika

Mérnöki optimalizáció

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Végeselem-analízis

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Faculty of Mechanical Engineering

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CAD BOOK

Course bulletin

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computer aided design, CAD, virtual model, solid model, shape feature, parametrical modeling, sheet metal parts, constraints, surface modeling, CAx systems

SUMMARY:

The activity of mechanical engineers has been changed during the last decade because the computer became an integral part of the design process. The engineers can solve more complex tasks with more effective and productive work. The aim of this coursebook is to overview the basis of computer aided engineering activity. The certain chapters deal with 3D solid model building techniques through the geometrical description till the features based parametric description. The material acquaints with modeling of sheet metal parts and surface modeling as well. The assembly modeling describes the various types of constraints and assembly operation procedures. An introduction to final elements method gives chance to insight of CAD numerical techniques. The CAD Book is dedicated to computer aided design, however all the CAx technology has a great importance for every engineering activity.

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Content

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CAD Book

1. Introduction

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The evolution of the informatics has increasing influence in every field of the our life, so the engineering is not mean exception. The work of engineers is changing, we can solve more complex problems, but the different software tools ensure effective and productive work.
The *CAD book* presents the topic of computer aided design (CAD) in the viewpoint of mechanical engineering, however the CAx technology has great importance in every engineering field.

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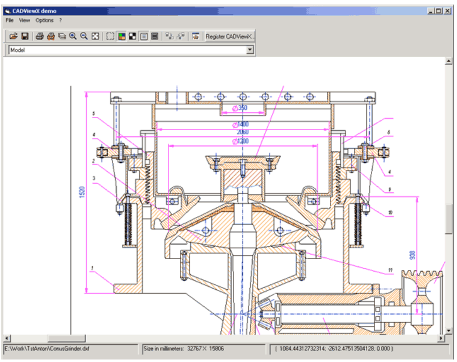
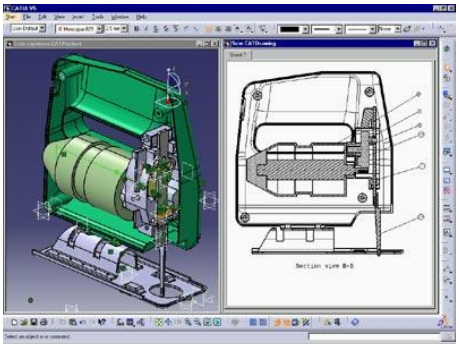
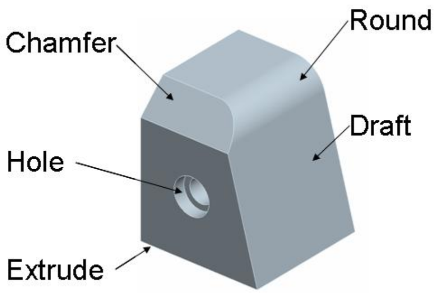
CAD – computer aided design







The CAD is the abriviation of Computer Aided Design, which means a wild range of computer software tools, which support the design process. A CAD system can be a simple 2D drawing system or a parametric associative hybrid modelling system.

The up-to-date method is this last concept, where

- the parametric means the dimension driven modelling,
- the associative means the live connection between the geometric elements,
- the hybrid means the parallel and synergic surface and solid modelling.

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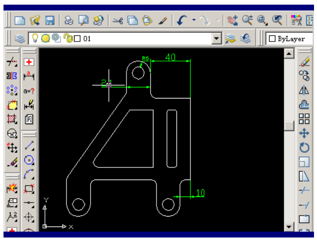
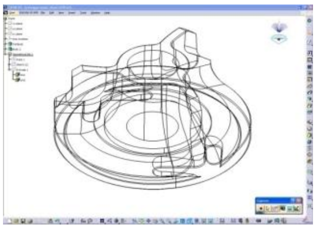
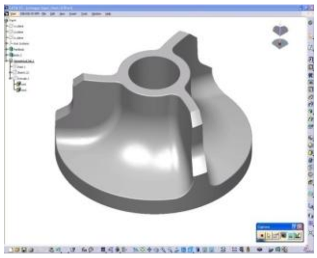
Classification of CAD systems

- **Application area**
- **Type of modelling**
  - 2D
  - 3D
- **Type of objects**
  - wire frame
  - surface
  - solid
  - hybrid
- **Parametrization**
  - Non-parametric
  - Parametric







The CAD systems can be classified by several viewpoint.

- The first is the application field. The CAD systems are developed in every industrial areas, so we can find systems in the field of mechanical engineering, electric engineering, architectural design, civil engineering, cloth and shoe design, medical application.
- The type of the modelling can be 2D, when the representation of the part is similar to the engineering drawing. The other method is the 3D modelling, when the model of the part is build in the virtual space.
- The applied modelling method can be wireframe modelling, when only the edges of the part are defined. In case of surface modelling the CAD model is hollow, only the boundary „skin" is defined. The solid modelling ensures realistic representation, the model consists of simple elementary elements.
- In case of parametric model, the size of the model is driven by the geometric parameters. The size of a non-parametric model is defined by user's modelling activity and the dimensioning value is driven by the modelled object.

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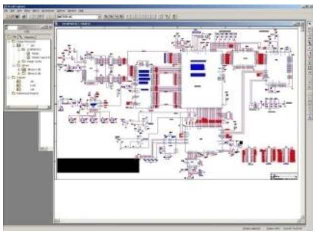
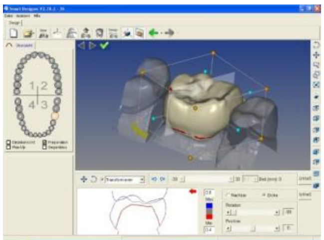
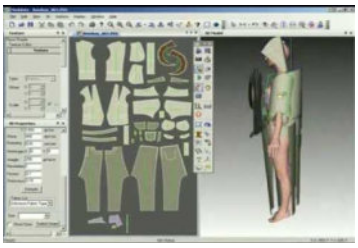
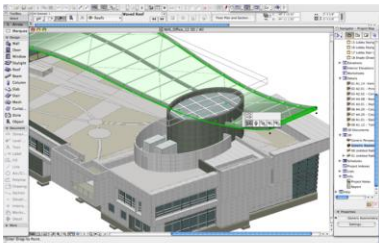
Areas of application

- Mechanical engineering
- Electronic design
- Architectural design
- Civil engineering
- Textile industry
- Medical









The pictures shows the most important application of CAD systems. The CAD systems were developed for these special application areas.

The typical application fields are:

- Mechanical engineering
- Electronic design
- Architectural design
- Civil engineering
- Textile industry
- Medical

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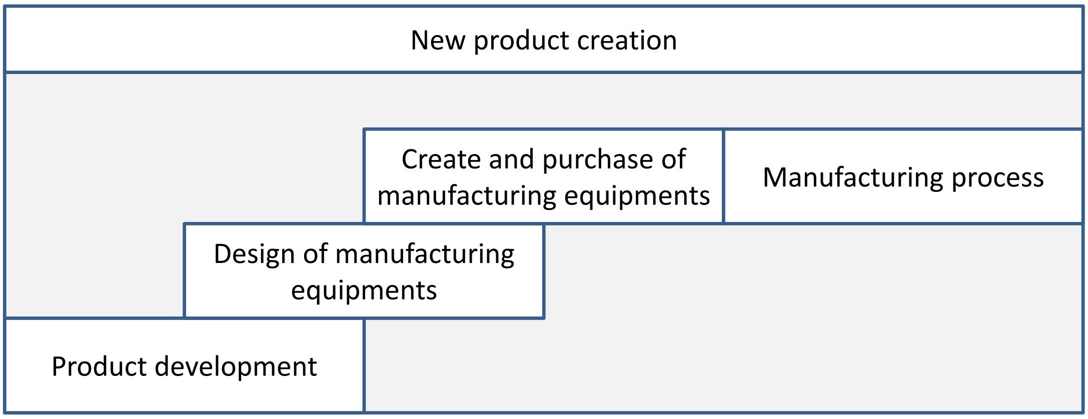
Process of new product creation



The new product creation process consists of four main step.

- The first is the product development, when the full design documentation is produced based on the market, customer and financial requirements.
- The production needs manufacturing equipments, like tools, machine tools, moulds etc. And if there are no exist, we have to design them. Then the manufacturing equipments have to purchase or create, which sometimes need lot of time and it has a high cost.
- The last stem is the production, which means part production and assembly.
- As the figure shows, some sub-processes can be performed with overlapping in order to reduce the lead time.

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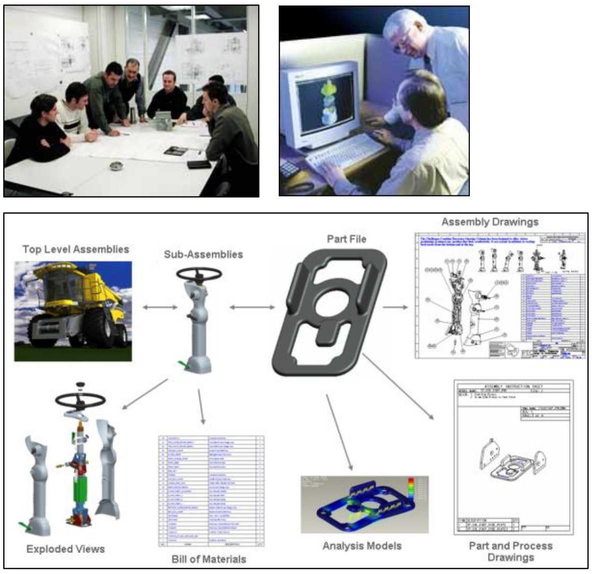
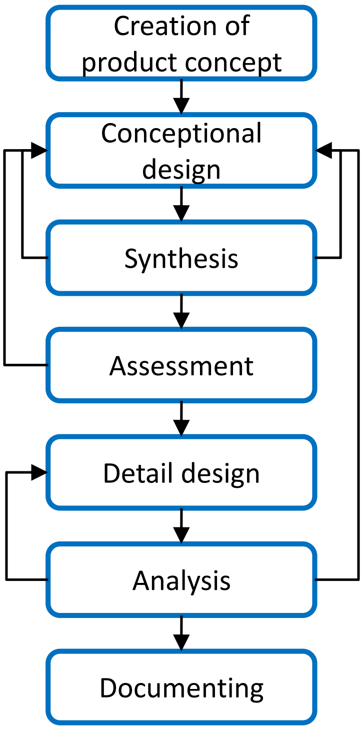
Process of product development





The steps of the product development are the next in general case:

- Creation of product concept. The function, engineering, quality, market and other requirements are collected in order to define the aim of the development.
- Conceptional design. The possible solution of each requirements are summarized.
- Synthesis. Unite the separated elements.
- Design assessment. The result is investigated in order to check, than it is suitable for the initial requirements.
- Detail design. The details of the product are designed.
- Analysis of the design. The product design is complete for analysis and every of important properties can be tested.
- Documenting. The result of the design process is the full design documentation.

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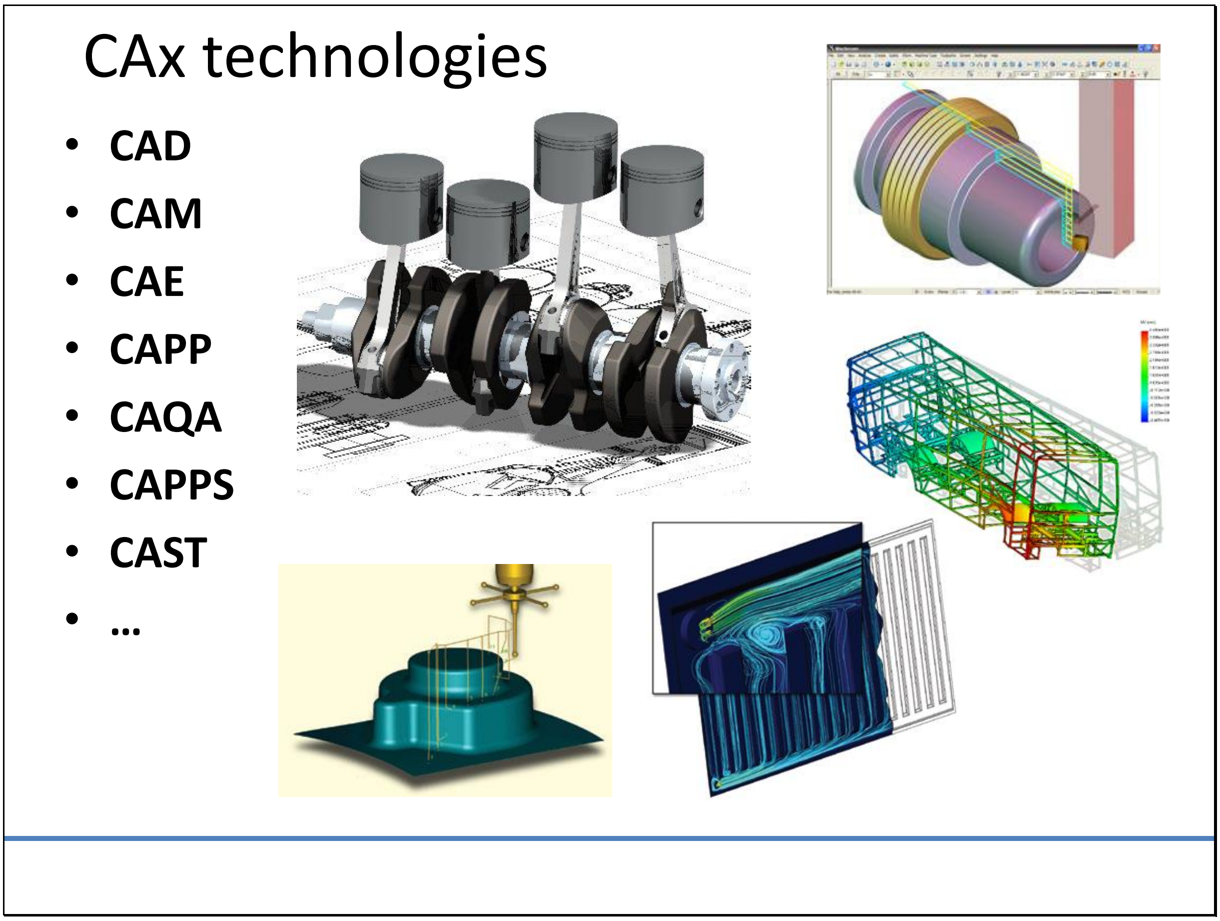

The product development and production process is supported by computer software. The name of this technology is CAx – computer aided something. These software tools support the specific engineering activities. The help of the computer means different things. In case of manufacturing the CNC programs are generated by a CAM system, the CAE means the collection of every engineering analysis and calculation. The task of the CAPP is to generate a process plan for manufacturing. The CAQA is the programming of coordinate measurement machines in general.

The most often used abbreviations are the next:

- CAD – computer aided design
- CAM – computer aided manufacturing
- CAE – computer aided engineering
- CAPP – computer aided process planning
- CAQA – computer aided quality assurance
- CAPPS – computer aided production planning and scheduling
- CAST – computer aided storage and transport

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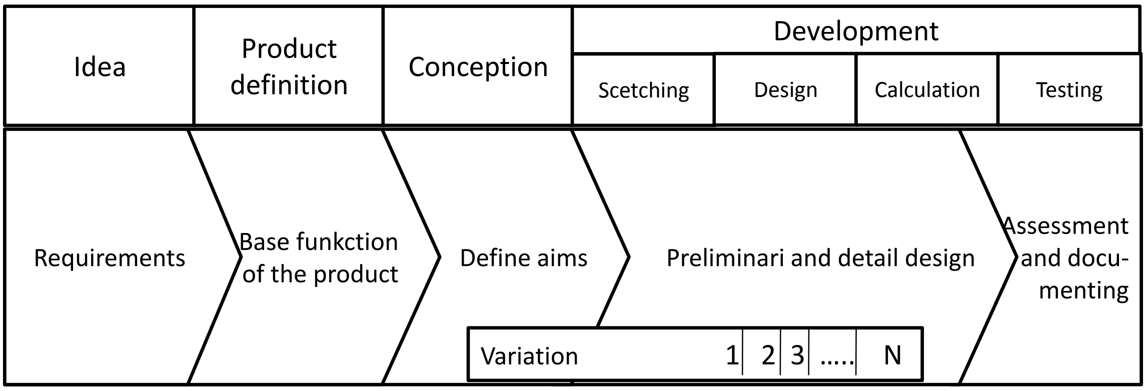
Product development (VDI 2221.)



The picture shows the steps of the product development process based on VDI 2221. recommendation.
The steps are similar as the previously mentioned process. The feedbacks and different product variations and the testing and assessment process has a great role during the product development. These activities characterise the lead time of the development.

The lead time will reduce if

- We have a clear product concept in the early phase of the development.
- The sub-processes can be performed parallel.
- The information flow is good, every member of the development have actual and up-to-date information.
- Use CAx tool.
- The motivation of the members are suitable.

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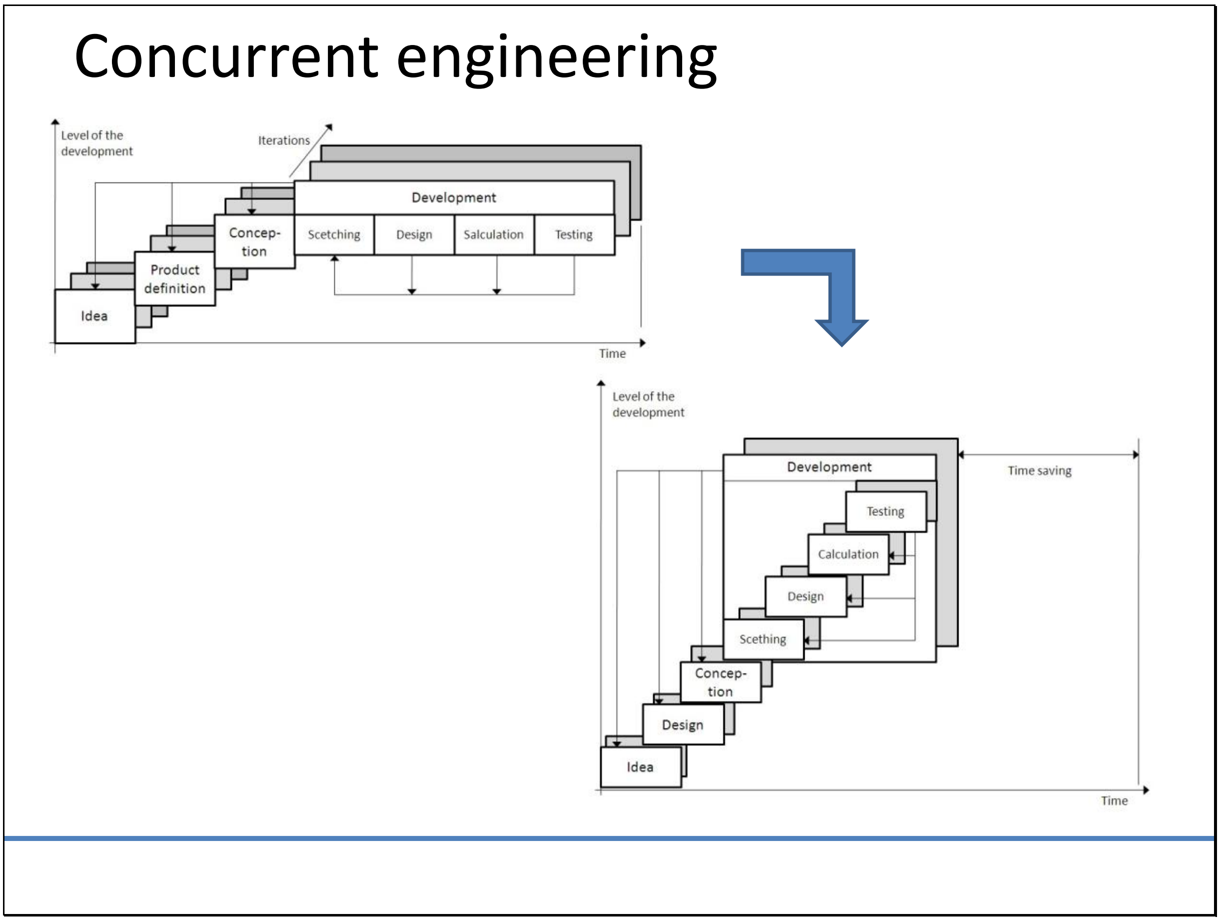

The steps of the product development were sequential, but in order to reduce lead time some activities are performed parallel with overlapping. This method is the simultaneous or concurrent engineering.

The application of the method needs

- Clear design process,
- High level collaboration between the members of the design team,
- Application of CAx systems,
- Common database.

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Optimal product

Production

Using

Recycling

The result of the development process is the product, which should be optimal solution of the initial requirements.

A product is optimal if it is

- suitable for production (material, manufacturing, assembly, inspection)
- Suitable for using (working, operation, safety etc.)
- Suitable for recycling.

During the inspections these viewpoints should be focused. The inspections and tests could be performed on prototype, final part of a model.

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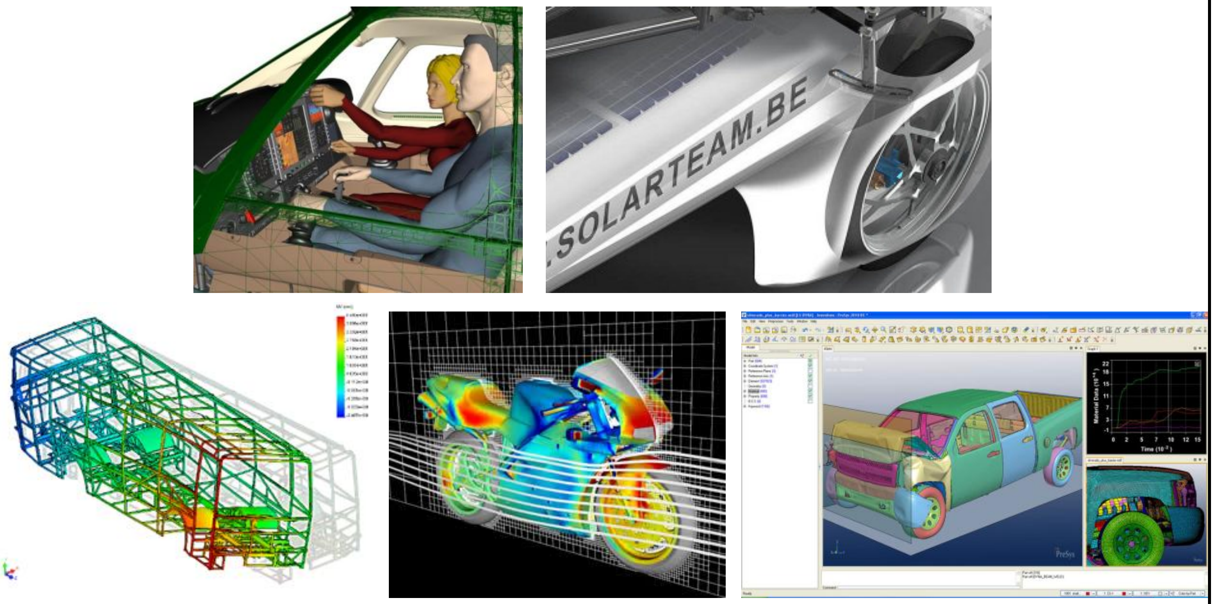
Simulation

imitation of the behaviour of a system



The simulation is the imitation of the behaviour of a system.

The simulation has an important role during the engineering design. The main roles of the product simulation:

- assessment of design alternatives,
- study the effect of the product to the environment,
- study the performance of the product during the use of it,
- investigation of the interaction between the product and the user.

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Prototype

Proof of concept Prototype
Form Study Prototype
User Experience Prototype
Visual Prototype
Functional Prototype

There is no general agreement on what constitutes a "prototype" and the word is often used interchangeably with the word "model" which can cause confusion. In general, "prototypes" fall into five basic categories:

A *Proof of concept prototype* is used to test some aspect of the intended design without attempting to exactly simulate the visual appearance, choice of materials or intended manufacturing process.

Form Study Prototype (Model) will allow designers to explore the basic size, look and feel of a product without simulating the actual function or exact visual appearance of the product. They can help assess ergonomic factors and provide insight into visual aspects of the product's final form.

User Experience Prototype (Model).

A *User Experience Model* invites active human interaction and is primarily used to support user focused research. While intentionally not addressing possible aesthetic treatments, this type of model does more accurately represent the overall size, proportions, interfaces, and articulation of a promising concept.

Visual Prototype (Model) will capture the intended design aesthetic and simulate the appearance, color and surface textures of the intended product but will not actually embody the function(s) of the final product. These models will be suitable for use in market research, executive reviews and approval, packaging mock-ups, and photo shoots for sales literature.

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Functional Prototype (Model) (also called a working prototype) will, to the greatest extent practical, attempt to simulate the final design, aesthetics, materials and functionality of the intended design. The functional prototype may be reduced in size (scaled down) in order to reduce costs. The construction of a fully working full-scale prototype and the ultimate test of concept, is the engineers' final check for design flaws and allows last-minute improvements to be made before larger production runs are ordered.

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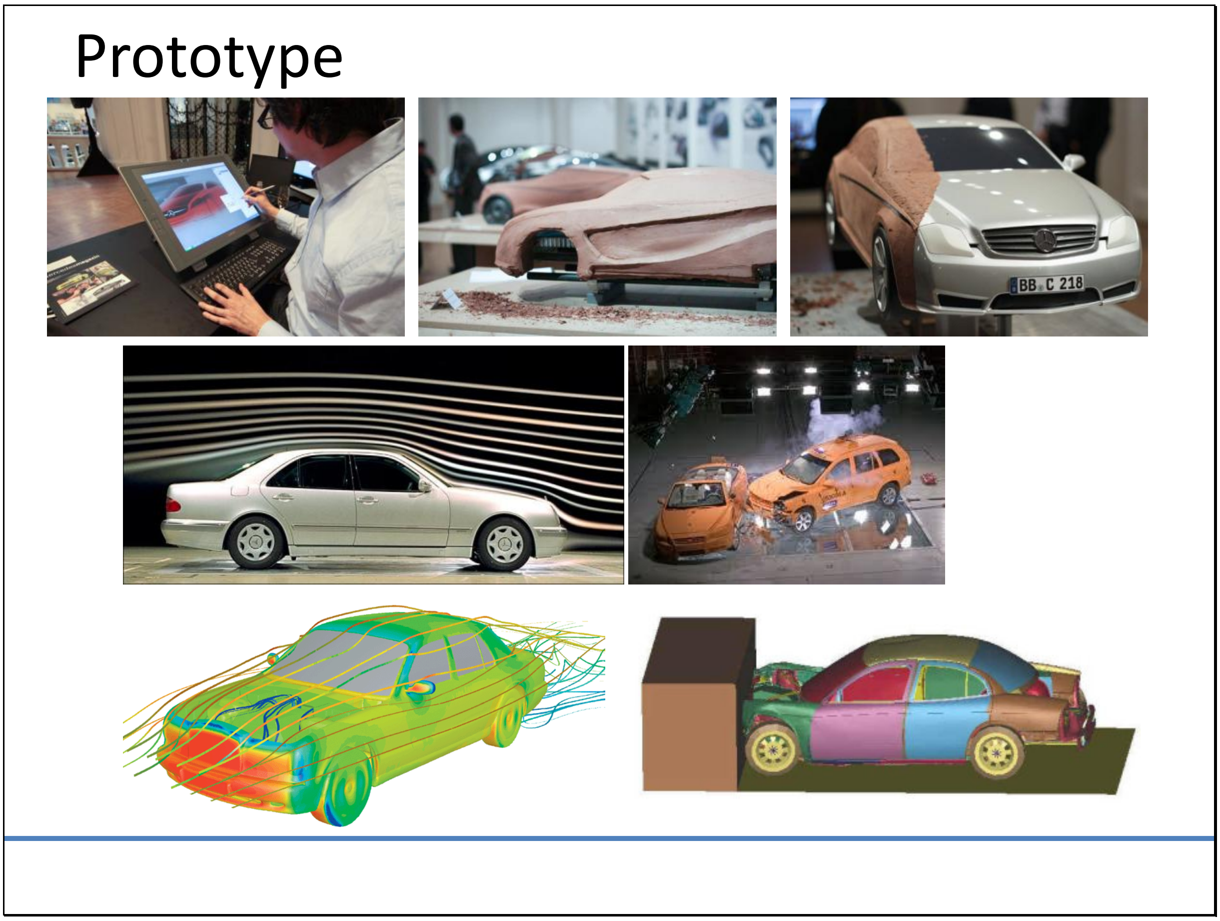

Differences between a prototype and a production design in general, prototypes will differ from the final production variant in three fundamental ways:

Materials: Production materials may require manufacturing processes involving higher capital costs than what is practical for prototyping. Instead, engineers will attempt to substitute materials with properties that simulate the intended final material.
Processes. Often expensive and time consuming unique tooling is required to fabricate a custom design. Prototypes will often compromise by using more variable processes, repeatable or controlled methods; substandard, inefficient, or substandard technology sources; or insufficient testing for technology maturity.
Lower fidelity. Final production designs often require extensive effort to capture high volume manufacturing detail. Such detail is generally unwarranted for prototypes as some refinement to the design is to be expected. Often prototypes are built using very limited engineering detail as compared to final production intent, which often uses statistical process controls and rigorous testing.

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CAD history

1960

1957 - Dr. Patrick J. Hanratty – *PRONTO* (the 1st CAM system)

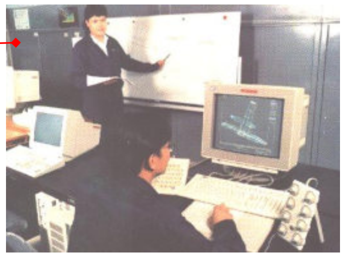
Early 1960's - Ivan Sutherland – *Sketchpad*

1965 - Dr. Hanratty, General Motors - *DAC* (Design Automated by Computer)

1966 - McDonnal-Dougles – *CADD*

1967 – Ford – *PDGS*

1967 – Lockheed – *CADAM*



In the early 60's the first CAD systems appeared, the origins of the development were automotive and aircraft industry. The 3D modelling mean wireframe modelling at this time. The mathematical fundaments of computer aided geometry were researched, the mathematical description of 3D curves and surfaces were created by de Casteljau and Bézier.

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CAD history

1970

Focus to the wide range of application

1975 - Avions Marcel Dassault – *CATIA*
(Computer Aided Three Dimensional
Interactive Application)

1980

1980 – DEC MicroVAX
1980 - IGES - Initial Graphic Exchange Standard

In the early 70's the development of the first CAD/CAM systems are closed and the industry focused to the wide range of application. The serious automotive and aircraft companies (Ford, General Motors, Mercedes-Benz, Toyota, Lockheed, McDonnell-Douglas) were the primary users and developers, these companies develop special systems for in-house application.

At 1975 the first 3D modelling CAD system was published by Avons Marcel Dassault. This was the CATIA: Computer Aided Three Dimensional Interactive Application.

In 1980 the DEC MicroVAX was the first computer, which didn't required special air-condition and electric connection.
The standard IGES file format was the first step to the cooperation between CAD systems and it started the integration and collaboration, which is the characteristic direction of the today's developments.

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CAD history

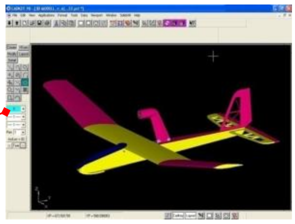
1980

1980 – IBM PC

1981 – Autodesk - AutoCAD Release 1
1982 - CADRA 2D CAD

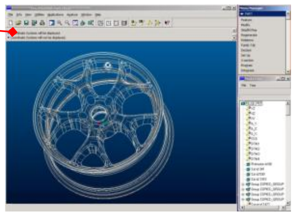
1984 - Bentley Systems – MicroStation
1984 - Diehl Graphsoft – *miniCAD*

1985 - Micro-Control System – *CADKEY*



1985 – CATIA v2

1987 - Parametric Technology - Pro/Engineer



1990

1989 – Unigraphics – *UniSolids*

The real revolution in informatics started with the IBM PC, because the hardware became cheaper and easy to use and work. The new generation of CAD systems utilised the advantages of the PC-s. The AutoCAD became the leader system in this decade.

The Pro/Engineer introduced the model-tree, which shows the history of the modelling process.

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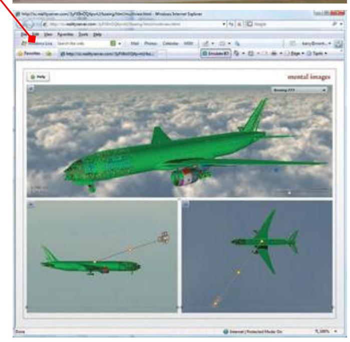
CAD history

1990

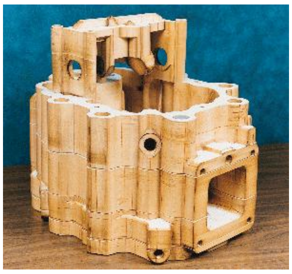
1990 – Boeing 777 – full CAD design process

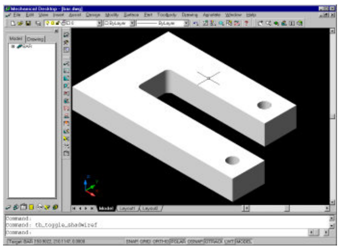


1990-94 - Autodesk AutoCAD
1 million licences / 4 years

1994 - Autodesk Mechanical Desktop 1.0

Commercial Rapid Prototyping Technologies





1995

In 1990 the Boeing 777 was the first project, which was performed by 3D CAD system. This milestone proved the justification of existence of 3D CAD systems and generated a new design process principles.

Rapid prototyping technologies appeared in the market, and the commercial systems ensured the rapid production of the physical prototypes.

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CAD history

1995

1995 - Intel Pentium Pro processor

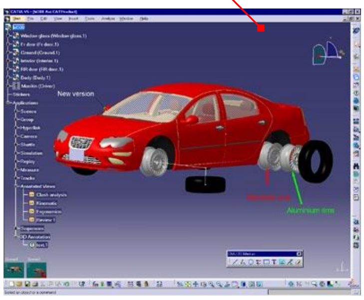
1996 - Windows NT operation system

1997 – OpenGL graphic card

1998 - Dassault System – *ENOVIA*
PDM - Product Data Management

1999 - Dassault System - **CATIA v5**

2000



In the second half of this decade the PC technology was renewed:

- The Intel Pentium Processor ensured the fast computing,
- The Windows NT ensured the effective multitasking, and
- The OpenGL technology ensured the fast computing of the 3D graphics.

The new trend in the integration was the PDM – Product Data Management, which extended the limits of the collaboration .

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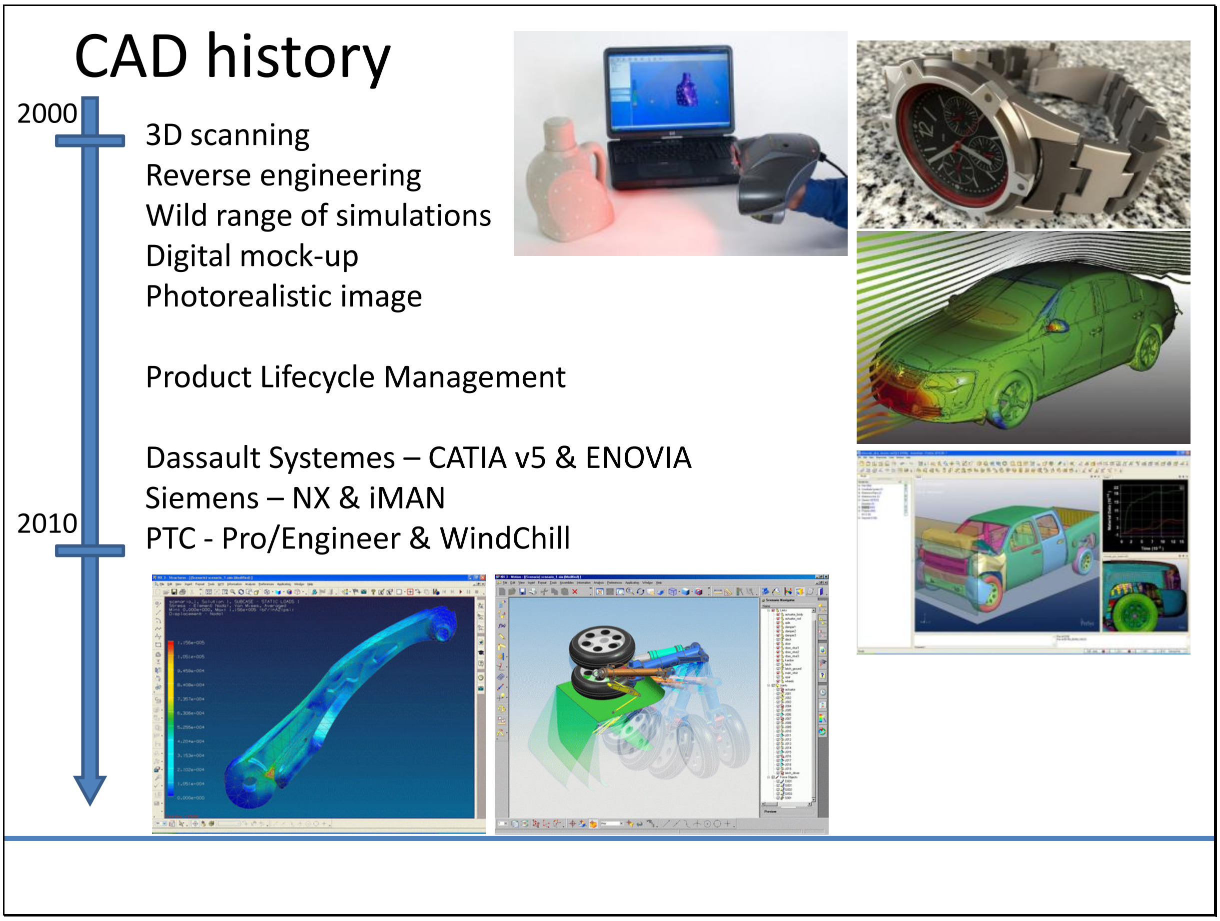

The prime mover of the development was the fast revolution of the PC hardware in the new century.

The main keywords of this decade are:

- 3D scanning and Reverse engineering: digital reproduction of the real parts
- Digital mock-up: digital prototype with wild range of simulations
- Photorealistic image
- Product Lifecycle Management: extended collaboration

The leader integrated CAD systems are

- The Dassault Systemes's CATIA v5 & ENOVIA
- The Siemens's NX & iMAN
- The PTC's Pro/Engineer & WindChill

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Hardware

Special devices for CAD applications

The evolution of the hardware can be presented by two examples:

- The IBM 7094 type computer (1970) needed special environment, many operator for maintenance, and large space. Nowadays a commercial laptop is able to serve CAD systems.

- In 1980 an IBM 3380 hard disk was 2.000 kg, the price was 800.000,- $, and the capacity was 20GB. In 2010. a microSD card is 1 g, the price is less than 100,- $, and the capacity is 32GB.

There are several special device for more effective use of CAD systems, like virtual reality helmet, special mouse and interactive modelling devices.

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CAD Book

2. Geometric fundaments of CAD systems

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GEOMETRIC ELEMENTS

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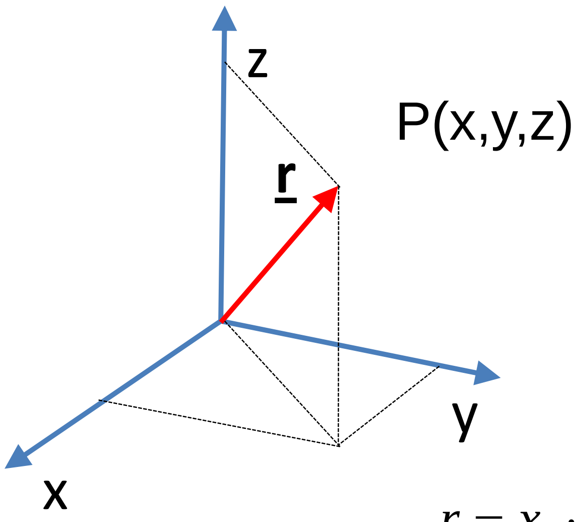
Point



$$\underline{r} = x_1 \cdot \underline{i} + x_2 \cdot \underline{j} + x_3 \cdot \underline{k} = \begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix}$$

In a CAD system in the 3D virtual space the geometric elements are represented in a Descartian coordinate system by x, y, and z values.
The simplest geometric element is the point, which is used as datum elements in a CAD modelling. The representation of a point is done by the 3 coordinate value.

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Curves

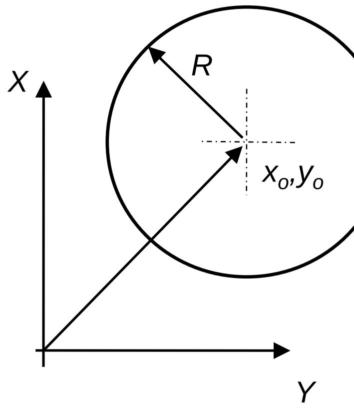
Explicit definition:

$x=x(t)$
$y=y(t)$
$t \in [0,1]$

Implicit definition:

$f(x,y)=0$

Example: CIRCLE



$$x = x_o + R \cdot \cos 2\pi t$$

$$y = y_o + R \cdot \sin 2\pi t$$

$$t \in [0,1]$$

$$(x - x_o)^2 + (y - y_o)^2 - R^2 = 0$$

A curve is a continuous set of points. A curve can be defined by explicit or implicit definition. The explicit formula is suitable for generating the points of the curve, and the implicit formula is suitable for investigating a location of a point. If the value of the formula is 0, the given point is the part of the curve.
In the CAD practice the explicit definition is applied.
The example shows the definition of a circle. The radii of the circle is R and the centre point is x_o, y_o.

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3D curves

General description by polinoms:

Explicite definition:

x=x(t)
y=y(t)
z=z(t)

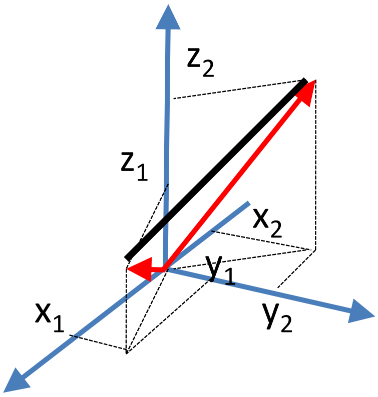
$$x(t)=\sum_{i=0}^{n} a_i \cdot t^i$$

$$y(t)=\sum_{i=0}^{n} b_i \cdot t^i$$

$$z(t)=\sum_{i=0}^{n} c_i \cdot t^i$$

$$t \in [0,1]$$

Example: LINE



$$x = x_1 \cdot t + x_2 \cdot (1-t)$$
$$y = y_1 \cdot t + y_2 \cdot (1-t)$$
$$z = z_1 \cdot t + z_2 \cdot (1-t)$$
$$t \in [0,1]$$

The 3 D curves can be defined by explicit formula. The example shows the definition of a line, which go through *(x_1,y_1,z_1)* and *(x_2,y_2,z_2)* points.
The classic curves, like line, circle, ellipse etc.) have explicit definition, but a general curve hasn't got a description. These curves can be defined by polynoms, which are adjusted by a_i, b_i, c_i factors. The polynoms can be differentiated continuously, which is essential for many investigation.

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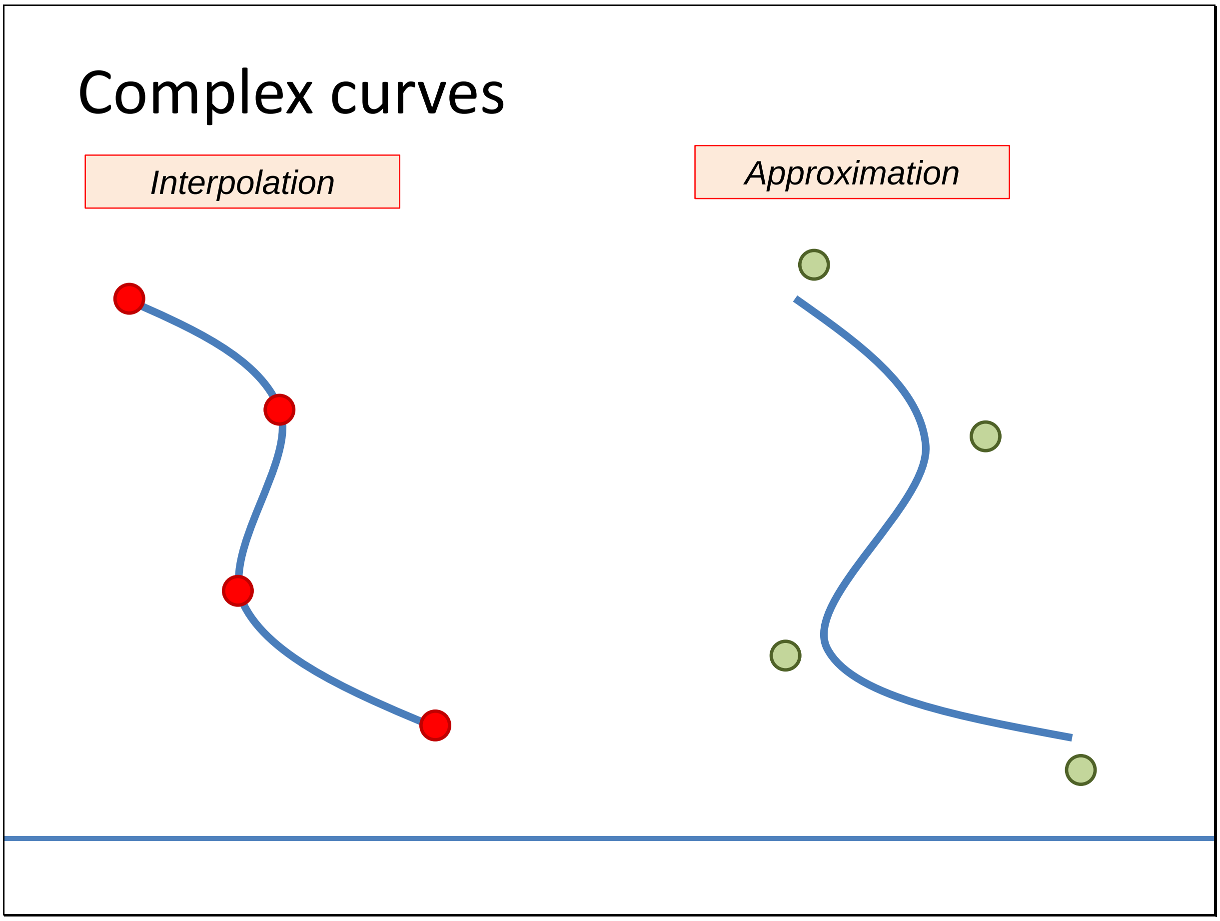

The set of factors are not so easy, therefore we use control points in the CAD environment in order to define a curve. We can speak about interpolation, if the curve goes through these points, or approximation, if the curve draws near to these points. Both of these methods are used in theoretic mathematic description and in CAD systems.

A complex curve can be defined by many points. We can use two strategy:

- Use a high degree polynom, or
- Multisegment low degree polynoms.

The high degree polynoms sometimes become wave, therefore we prefer the second way of curve design. The connecting segments have to be continuous, and the continuity has different aspects.

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Lagrange interpolation

$$\vec{r}(t_j) = \left[x(t_j), y(t_j)\right] = \sum_{i=0}^{n-1} [a_i, b_i] \cdot t_j^i = \vec{r}_j$$

$$\vec{r}(t) = \sum_{i=0}^{n-1} L_i(t) \cdot \vec{r}_i$$

$$L_i(t) = \frac{\prod_{j \neq i} (t - t_j)}{\prod_{j \neq i} (t_i - t_j)}$$

In case of Lagrange interpolation the control points are **r**$_1$, **r**$_2$,… **r**$_n$. We found the minimum degree L(t) polynom, which gives **r**$_1$, **r**$_2$,… **r**$_n$ points in t_1, t_2, … t_n. The number of degree of the polynom will be (n-1), and the required [a_i,b_i] factors can be calculated from the equation system, which gives from the j=1, 2, … n points. The result is the $L_i(t)$ weight function.

In case of modification of one point, it has an influence to the whole polynom and this is the main disadvantage of the Lagrange interpolation polynom.

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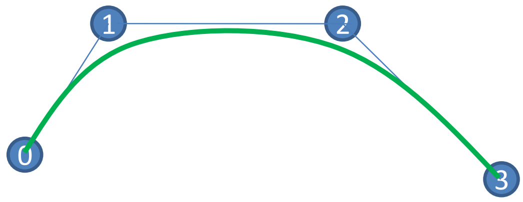
Bézier interpolation

(1) $(t+(1-t))^n = \sum_{i=0}^{n} \binom{n}{1} \cdot t^i \cdot (1-t)^{n-i}$ $\quad B_i^{(n)}(t) = \binom{n}{i} \cdot t^i \cdot (1-t)^{n-i}$

(2) $\underline{b}(t) = \sum_{i=0}^{3} \underline{b}_i \cdot \binom{3}{i} \cdot t^i \cdot (1-t)^{3-i}$

$$\underline{b}(t) = \underline{b}_0 \cdot (1-t)^3 + 3 \cdot \underline{b}_1 \cdot t \cdot (1-t)^2 + 3 \cdot \underline{b}_2 \cdot t^2 \cdot (1-t) + \underline{b}_3 t^3$$

$$\underline{b}(t) = \underline{b}_o \cdot (-t^3 + 3 \cdot t^2 - 3 \cdot t + 1) + \underline{b}_1 \cdot (3 \cdot t^3 - 6 \cdot t^2 + 3t) + \underline{b}_2 \cdot (-3 \cdot t^3 - 3 \cdot t^2) + \underline{b}_3 t^3$$



The Bézier interpolation polynome is the most known polynom, which was created in 1972 for the CAD applications.

It uses control points:

- The curve will go through the outside points ($\mathbf{p}_o$ and $\mathbf{p}_n$),
- The tangent vector in the outside points are $\mathbf{p}_1$-$\mathbf{p}_0$ and $\mathbf{p}_{n-1}$-$\mathbf{p}_n$.
- The weight function should be symmetric, so the curve will be same if the order of the points will be changed.

The Bernstein polynome (1) is one of the result of this problem.

In case on n=3 Bézier curve needs four control points, the curve go through in 1st and 4th points, and draws neat 2nd and 3rd.

The cubic Bézier curve is the (2).

The tangent vectors in the start and finish points are:

$\underline{b}'(0) = 3\ (\underline{b}_1 - \underline{b}_0)$

$\underline{b}'(1) = 3\ (\underline{b}_3 - \underline{b}_2)$

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Spline

(1) $\vec{p}(t) = \vec{a}_3 \cdot t^3 + \vec{a}_2 \cdot t^2 + \vec{a}_1 \cdot t + \vec{a}_0$

(2) $\vec{p}(0) = \vec{a}_0$ $\vec{p}(1) = \vec{a}_3 + \vec{a}_2 + \vec{a}_1 + \vec{a}_0$

$\vec{p}'(0) = \vec{a}_1$ $\vec{p}'(1) = 3 \cdot \vec{a}_3 + 2 \cdot \vec{a}_2 + \vec{a}_1$

(3) $\vec{p}_i(0) = \vec{r}_i$

$\vec{p}_i(1) = \vec{r}_{i+1}$

$\vec{p}_i'(1) = \vec{p}_{i+1}'(0)$

$\vec{p}_i''(1) = \vec{p}_{i+1}''(0)$

The simplest polynom, which has a constant 2nd derivative is the cubic spline (1).
The conditions of the continuity is the equality of the p(t) and the p'(t) in the start and end points (2).
The parameters of the *i*th segments are identified by the (3), but there is several results, because the number of the unknown variables are higher then the number of equations.

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B-spline

$$\vec{r}(t) = B_0(t)\cdot\vec{r}_0 + B_1(t)\cdot\vec{r}_1 + B_2(t)\cdot\vec{r}_2 + B_3(t)\cdot\vec{r}_3$$

Weights:

$$B_0(t) = \frac{(1-t)^3}{6}$$

$$B_1(t) = \frac{1+3\cdot(1-t)+3\cdot t\cdot(1-t)^2}{6}$$

$$B_2(t) = \frac{1+3\cdot t+3\cdot(1-t)\cdot t^2}{6}$$

$$B_3(t) = \frac{t^3}{6}$$

If the value of the derivatives are defined in the start and end points, the equation can be solved. This is the B-spline.

The B-spline is

- an approximation curve, it doesn't go through the control points,
- the control points has not got any effects to the other segments.

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Surfaces

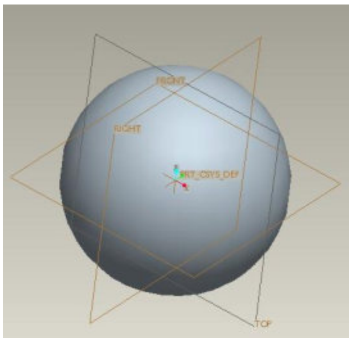
Explicit definition:

$x=x(u,v)$
$y=y(u,v)$
$z=z(u,v)$
$u,v \in [0,1]$

Implicit definition:

$f(x,y,z)=0$

Example: SPHERE



$$x = x_o + R \cdot \cos 2\pi u \cdot \sin \pi v$$

$$y = y_o + R \cdot \sin 2\pi u \cdot \sin \pi v$$

$$z = z_o + R \cdot \cos \pi v$$

$$u,v \in [0,1]$$

$$(x-x_o)^2 + (y-y_o)^2 + (z-z_o)^2 - R^2 = 0$$

The 3D surfaces can be defined by explicit and implicit equation, as the points or curves, but in case of explicit surface definition two parameters are used (u,v). The values of them are between 0 and 1.
The example shows the definition of a sphere, the centre of it is (x_o, y_o, z_o), the radii is R.

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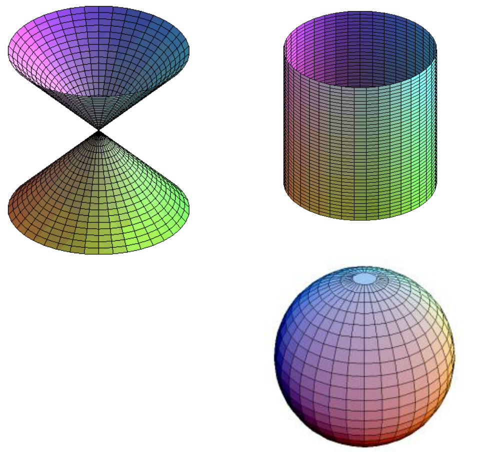
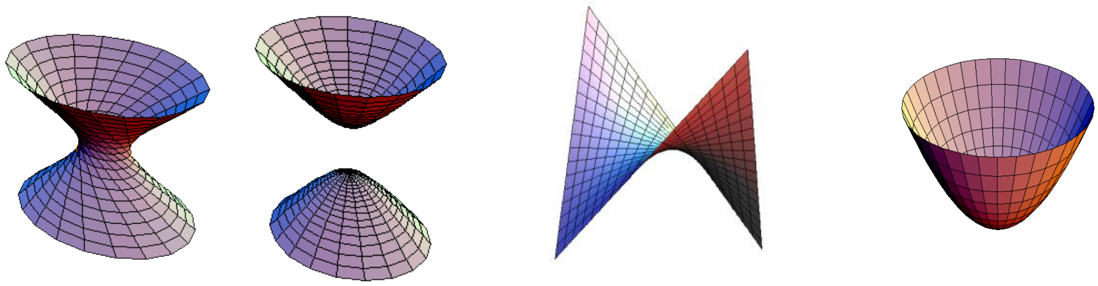
Quadratic surface

$$\begin{bmatrix} x & y & z & 1 \end{bmatrix} \cdot \underline{Q} \cdot \begin{bmatrix} x \\ y \\ z \\ 1 \end{bmatrix} = 0$$





If any parameters are quadratic, the surface call quadratic surface. These surfaces can be described by homogenous form, where the Q factor matrix is constant in case of each surfaces.
This format is suitable for describe sphere, cylinder, cone, hyperboloid, paraboloid etc.

These analytic surfaces are not suitable for describe the surfaces of a machine part, therefore we have to use complex and freeform surfaces in the CAD system. The three most known type of these surfaces are

- Sweep surface
- Ruled surface
- Freeform or sculpture surface.

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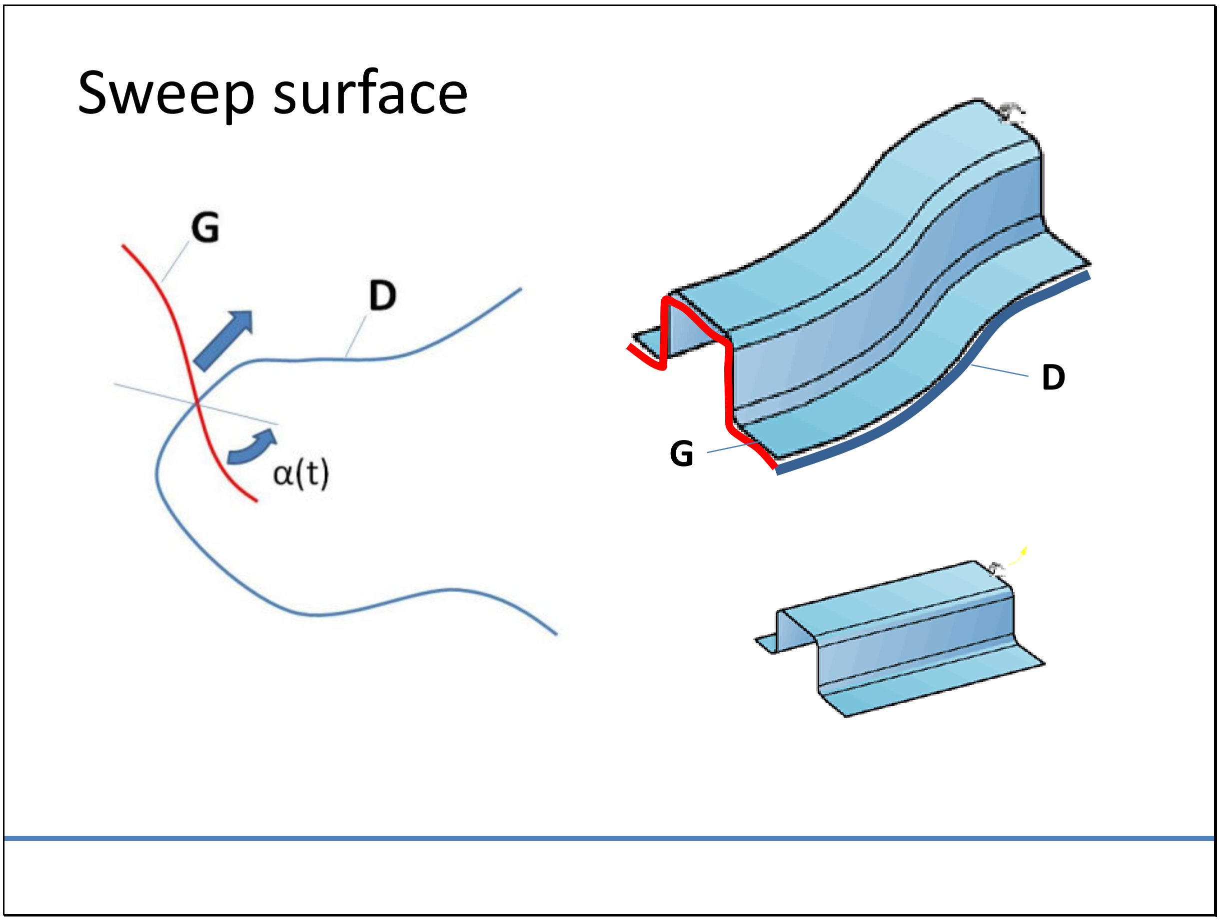

In case of sweep surface two curve have to defined.

- The first is an open or closed curve (D).
- The second curve (G) will run along D with the constant contact point.
- There is possible to use a rotation (α(t)) function.

The plane, sphere, cylinder, cone can be defined as sweep surface.

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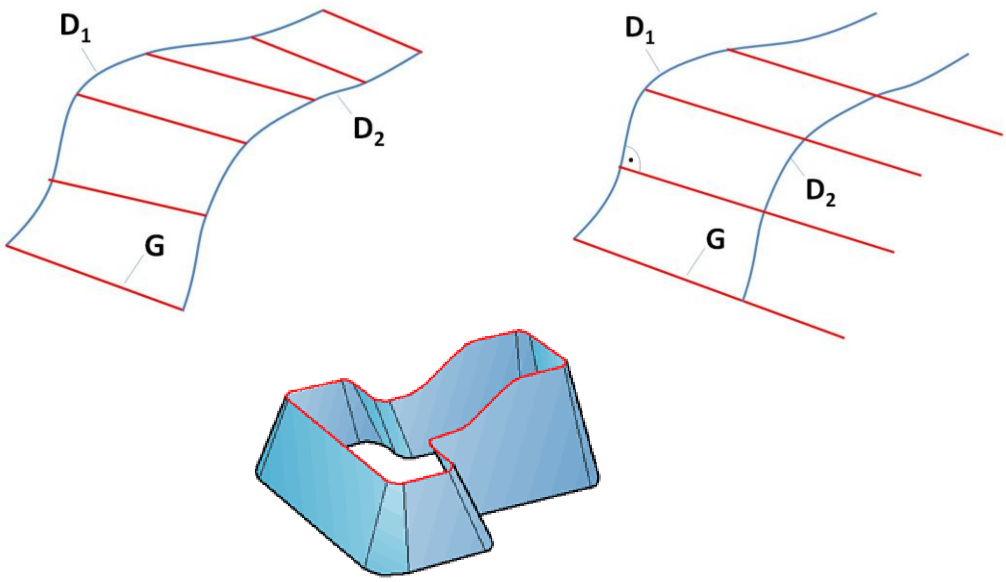
Ruled surface



The ruled surface is defined by three 3D curves.
The G curve drive along D1 curve and lean in D2.
In the first case the D1 and D2 are divided to equal segments, and the end points of these segments are connected by G.
In the second case the G curve just lean to D1, and the G will be parallel in every position.
Other variation can be generated of ruled surface by application of a non constant G curve.

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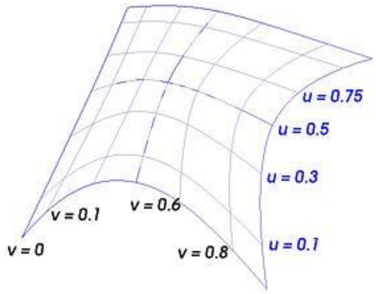
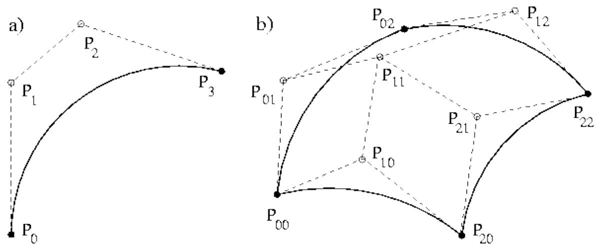
Freeform surfaces

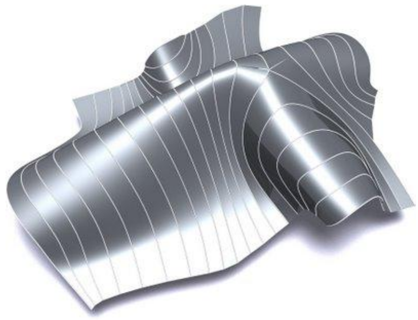
(1) $\vec{r}(u,v) \qquad u,v \in [0,1]$

(2) $\vec{r}(u,v) = \sum_{i=0}^{m}\sum_{j=0}^{n} \vec{r}_{ij} \cdot B_{ij}(u,v) \qquad u,v \in [0,1]$

(3) $B_{ij}(u,v) = \binom{n}{i} \cdot u^i \cdot (1-u)^{n-i} \cdot \binom{m}{j} \cdot v^i \cdot (1-v)^{m-j}$







If a surface cannot be describe by analytic or moving of curves, they are called freeform or sculpture surfaces. The mathematic presentation of these surfaces are similar to the spline curves, control points are used to determine the surface.

The parametric surface description uses two variables (u,v), and the surface is identified by weight functions (1)(2), like in case of curves.

The Bézier-surface uses Bézier curves as control geometry (3).

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TRANSFORMATIONS

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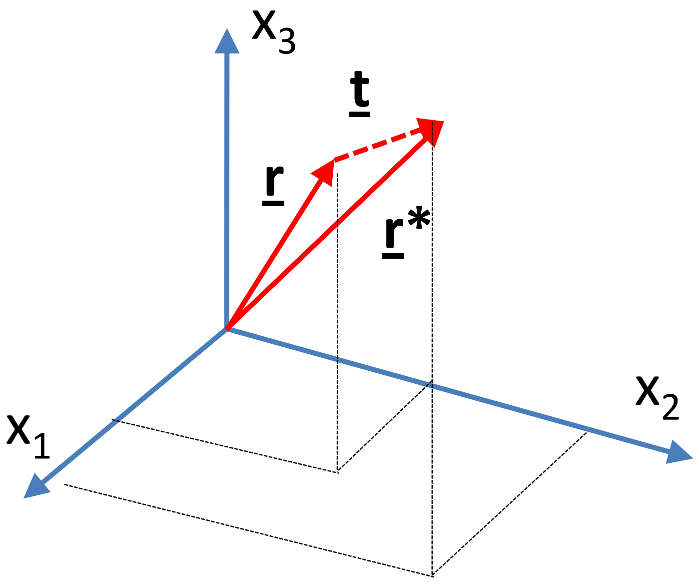
Translation

$$\underline{r}^* = \underline{r} + \underline{t} = \begin{bmatrix} x_1 + t_1 \\ x_2 + t_2 \\ x_3 + t_3 \end{bmatrix}$$

$$\underline{t} = \begin{bmatrix} t_1 \\ t_2 \\ t_3 \end{bmatrix}$$



$\underline{t}$: translation vector

The defined geometric elements should be modified or transformed in a CAD system. This transformation is done by point-by-point, so we have to understand the manipulation methods of a point.

The simplest transformation is the translation, when the point, which is represented with $\underline{r}$ vector, is moved by $\underline{t}$ vector.

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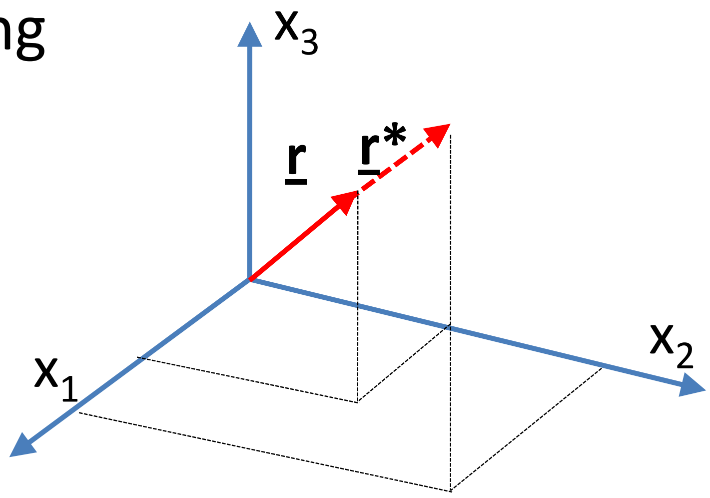
Scaling



C > 0

$$\underline{r}^* = \underline{C} \cdot \underline{r} = \begin{bmatrix} C & 0 & 0 \\ 0 & C & 0 \\ 0 & 0 & C \end{bmatrix} \cdot \begin{bmatrix} r_1 \\ r_2 \\ r_3 \end{bmatrix} = \begin{bmatrix} C \cdot r_1 \\ C \cdot r_2 \\ C \cdot r_3 \end{bmatrix}$$

In case of scaling, every coordinate values are multiple with a constant. These constants can be same, this is the uniform scaling, or these factors can be different. The scaling is calculated by matrix multiplication, where C is the scaling matrix.

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Rotation about x_i

Rotation around x_1 with φ_1 $\quad r^* = \underline{F}_1 \cdot \underline{r}$

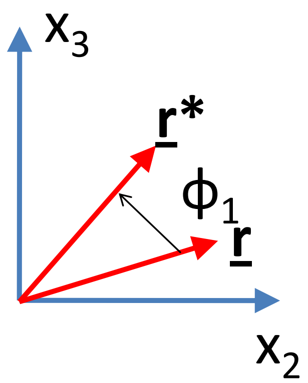
Rotation around x_2 with φ_2 $\quad r^* = \underline{F}_2 \cdot \underline{r}$

Rotation around x_3 with φ_3 $\quad r^* = \underline{F}_3 \cdot \underline{r}$

$$\underline{F}_1 = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\varphi_1 & -\sin\varphi_1 \\ 0 & \sin\varphi_1 & \cos\varphi_1 \end{bmatrix}$$

$$\underline{F}_2 = \begin{bmatrix} \cos\varphi_2 & 0 & \sin\varphi_2 \\ 0 & 1 & 0 \\ -\sin\varphi_2 & 0 & \cos\varphi_2 \end{bmatrix}$$

$$\underline{F}_3 = \begin{bmatrix} \cos\varphi_3 & -\sin\varphi_3 & 0 \\ \sin\varphi_3 & \cos\varphi_3 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$



$$\underline{r}^* = \underline{\underline{F}}_2\, \underline{\underline{F}}_1\, \underline{r}$$

The rotation of an object means the rotation around a x_i coordinate axes with a φ_i angle. If the rotation is performed around a general line, the coordinate system has to be transformed to the direction of the line.

The rotation is calculated by matrix multiplication, where $\underline{F}_i$ is the rotation matrix. The order of the multiplication is important if more rotations are applied.

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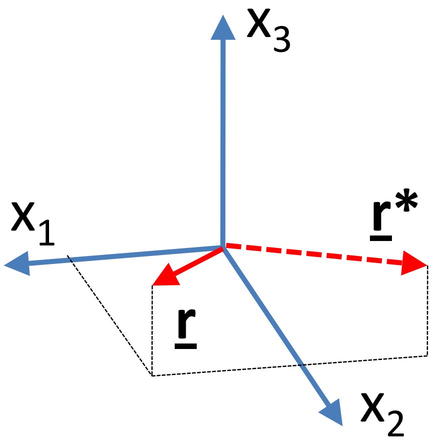
Mirror to plane



Mirror to [x_2, x_3] plane:	$\underline{r}^* = \underline{S}_1 \cdot \underline{r}$
Mirror to [x_1, x_3] plane:	$\underline{r}^* = \underline{S}_2 \cdot \underline{r}$
Mirror to [x_1, x_2] plane:	$\underline{r}^* = \underline{S}_3 \cdot \underline{r}$

$$\underline{S}_1 = \begin{bmatrix} -1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad \underline{S}_2 = \begin{bmatrix} 1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad \underline{S}_3 = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & -1 \end{bmatrix}$$

The mirror of an object has different ways. The first is the mirror to coordinate plane. We use matrix multiplications, as previous. $\underline{S}_i$ is the mirror matrix. The matrix is very simple, depends on the actual plane, the sign of appropriate coordinate value is changed.

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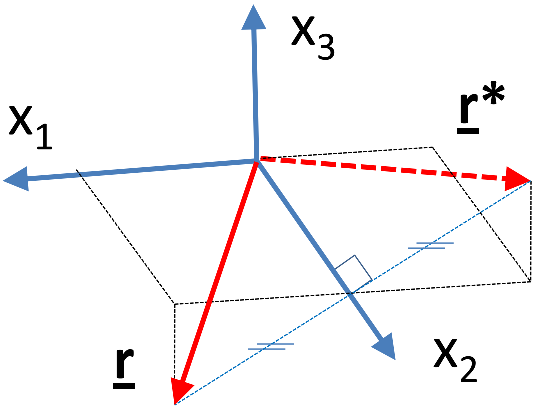
Mirror to x_i axes



Mirror to x_1 axes:

$$\underline{r}^* = \underline{S}_{2,3} \cdot \underline{r}$$

Mirror to x_2 axes:

$$\underline{r}^* = \underline{S}_{1,3} \cdot \underline{r}$$

Mirror to x_3 axes:

$$\underline{r}^* = \underline{S}_{1,2} \cdot \underline{r}$$

$$\underline{S}_{2,3} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & 0 & -1 \end{bmatrix} \quad \underline{S}_{1,3} = \begin{bmatrix} -1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & -1 \end{bmatrix} \quad \underline{S}_{1,2} = \begin{bmatrix} -1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

The second way is the mirror to x_i axes. As the $\underline{S}_{i,j}$ mirror matrix shows, the signs of the values of coordinate axis are changed, expect the x_i.

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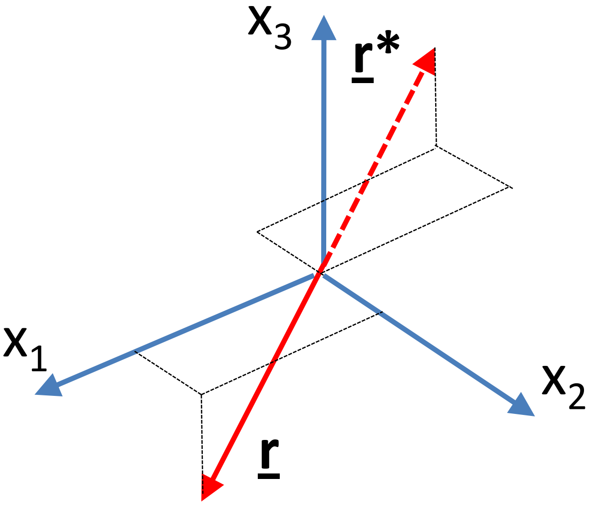
Mirror to the origin



$$\underline{r}^* = \underline{S} \cdot \underline{r}$$

$$\underline{S} = \begin{bmatrix} -1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & 0 & -1 \end{bmatrix}$$

The mirror to the origin is very simple, every sign of the coordinate values have to be changed. Therefore the mirror matrix contains -1 in the main diagonal.

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PROJECTION FOR DISPLAY

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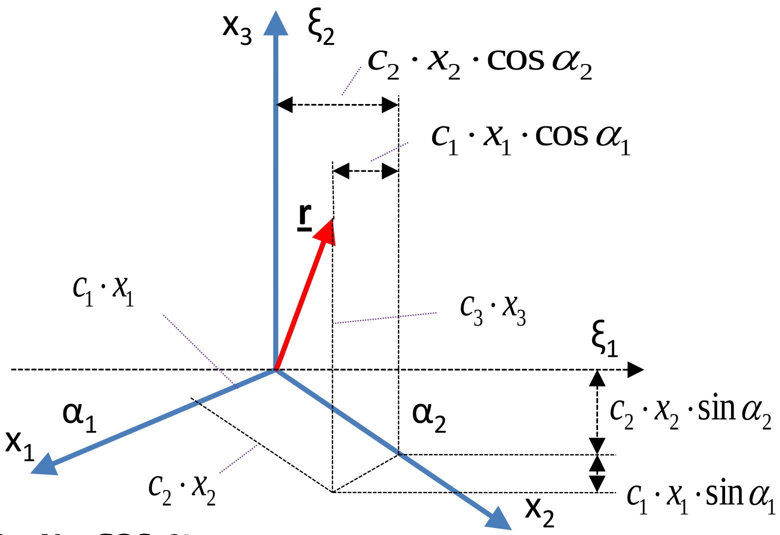
Projection

$$\begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} \Rightarrow \begin{bmatrix} \xi_1 \\ \xi_2 \end{bmatrix}$$



$$\xi_1 = c_2 \cdot x_2 \cdot \cos\alpha_2 - c_1 \cdot x_1 \cdot \cos\alpha_1$$

$$\xi_2 = c_3 \cdot x_3 - c_2 \cdot x_2 \cdot \sin\alpha_2 - c_1 \cdot x_1 \cdot \sin\alpha_1$$

$$\begin{bmatrix} \xi_1 \\ \xi_2 \end{bmatrix} = \begin{bmatrix} -c_1 \cdot \cos\alpha_1 & c_2 \cdot \cos\alpha_2 & 0 \\ -c_{1.} \sin\alpha_1 & -c_{2.} \sin\alpha_2 & c_3 \end{bmatrix} \cdot \begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix}$$

The core of the CAD system (kernel) compute the 2D coordinate values ($\underline{\rho}$) to the display the 3D object ($\underline{r}$). The connection between the two vectors is computed by matrix multiplication ($\underline{\rho} = \underline{A}\ \underline{r}$). The $\underline{A}$ matrix is the projection matrix.

Based on the picture the $\underline{A}$ matrix can be defined easy.

The c_1, c_2, c_3 factors show the scale on the x_i axes, the α_1, α_2 angles show the angles between the x_1, x_2 and ξ_1 axis.

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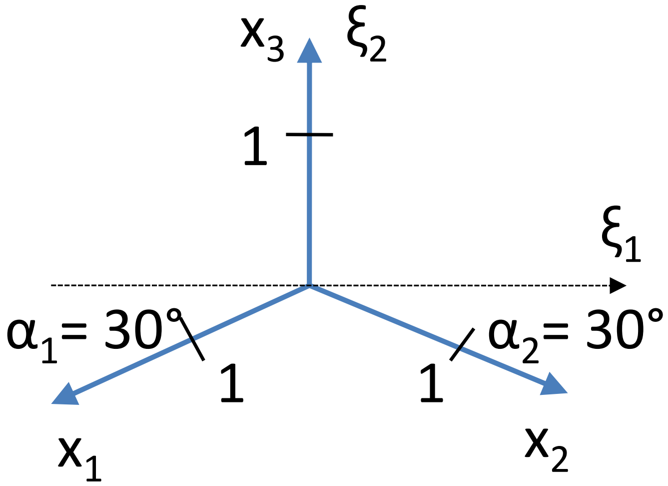
Isometric axonometry



$\alpha_1 = \alpha_2 = 30°$

$c_1 = c_2 = c_3 = 1$

$$\underline{A} = \begin{bmatrix} -\frac{\sqrt{3}}{2} & \frac{\sqrt{3}}{2} & 0 \\ -\frac{1}{2} & -\frac{1}{2} & 1 \end{bmatrix}$$

There are some special sets of c_i and α_i parameters, which are popular in the field of engineering image generation.

The first is called isometric axonometry, where, there is no scaling (c_i =1), and the position of the x_1 and x_2 axis are symmetric and the angles are 30° in both cases.

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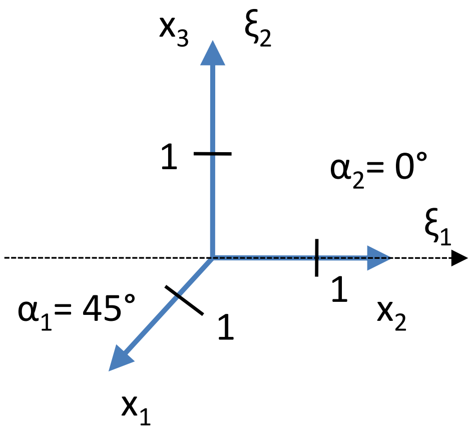
Frontal axonometry



$\alpha_1 = 45°$
$\alpha_2 = 0°$
$c_1 = ½$
$c_2 = c_3 = 1$

$$\underline{A} = \begin{bmatrix} -\frac{\sqrt{2}}{4} & 1 & 0 \\ -\frac{\sqrt{2}}{4} & 0 & 1 \end{bmatrix}$$

In case of frontal axonometry the x_2 axis is equal to the ξ_1 axis, there are no scaling in x_2 and x_3 axis, but the measures in x_1 are just half.
So the parameters are: $\alpha_1 = 45°$; $\alpha_2 = 0°$; $c_1 = 1/2$; $c_2 = c_3 = 1$.
In this case the front view of the part will be same as in 2D engineering drawing.

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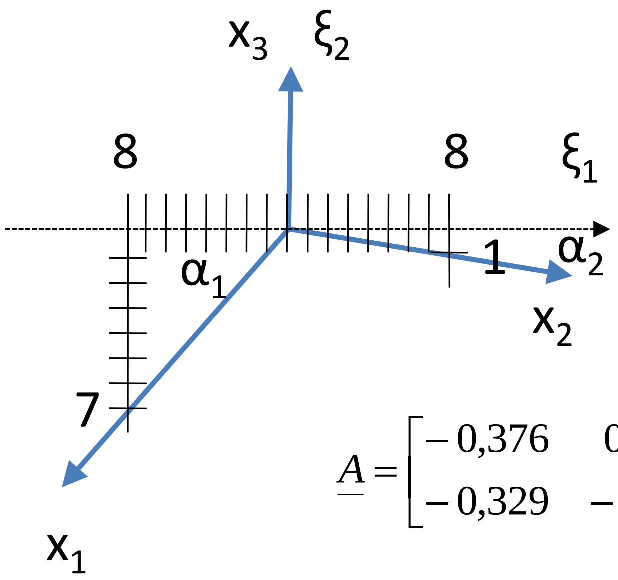
Dimetric axonometry



α_1 = arctg 7/8 = 41°10’
α_2 = arctg 1/8 = 7°10’
$c_1 = c_2 = c_3 = 1$

$$\underline{A} = \begin{bmatrix} -0,376 & 0,992 & 0 \\ -0,329 & -0,125 & 1 \end{bmatrix}$$

In case of dimetric axonometry there is no scaling on axis ($c_1 = c_2 = c_3 = 1$). The positions of the x1 and x2 are special (α_1 = arctg 7/8 = 41°10’, α_2 = arctg 1/8 = 7°10’).
The look of the part will be harmonic and natural.

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DISPLAY AND SHADING

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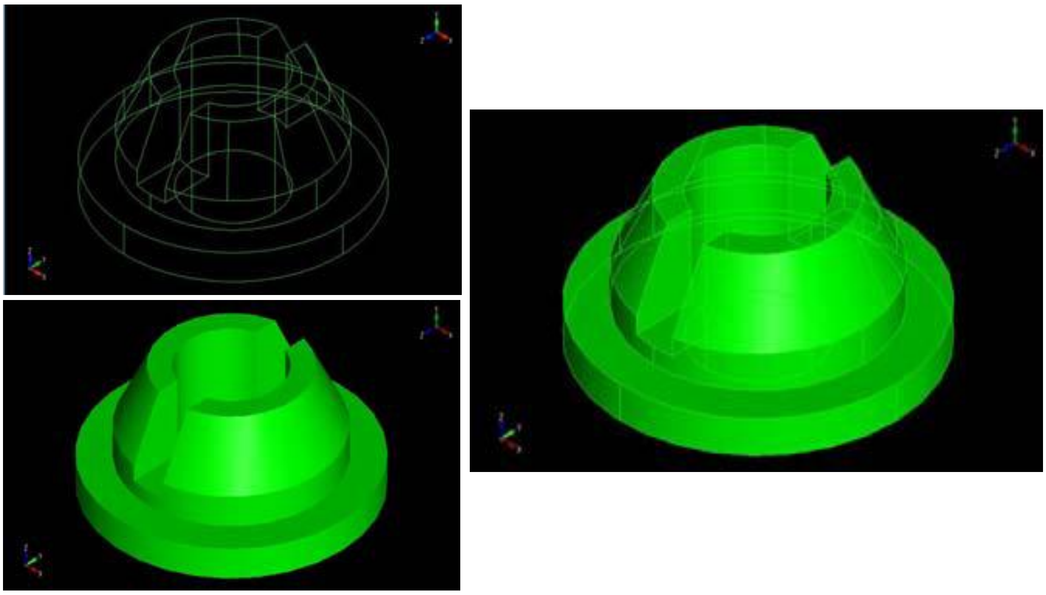
Wire-frame model, rendered and combined model



The model helps organize and visualize products and high level goals or activities. The model is complex and integrated. It would be the basic of analysis.

On the slide there are three types of display of the same model. Wire-frame model, rendered, realistic model and rendered model.

(http://en.wikipedia.org/wiki/Wire-frame_model)

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Hidden surface determination

Algorithms:

- Back face culling
- Z-buffer algorithm
- Raytracing
- Recursive Raytracing

The hidden surface determination is the a process which use to determine which surfaces or parts of surfaces are visible from a certain viewpoint. A hidden surface determination algorithm is a solution to the visibility problem. The analogue for line rendering is hidden line removal. Hidden surface determination is necessary to render an image correctly. So that one can't look through walls in virtual reality.

Some algorithm which can be use for this problem are the back face culling, the ray-tracing, the recursive ray-tracing of the Z-buffer algorithm.

(http://en.wikipedia.org/wiki/Hidden_face_removal)

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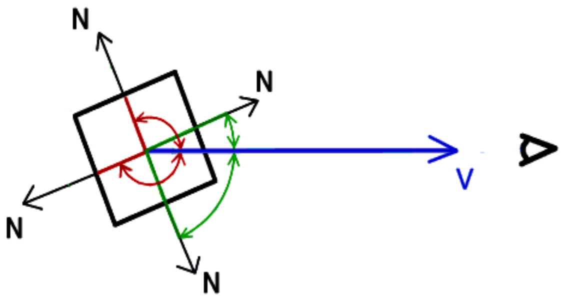
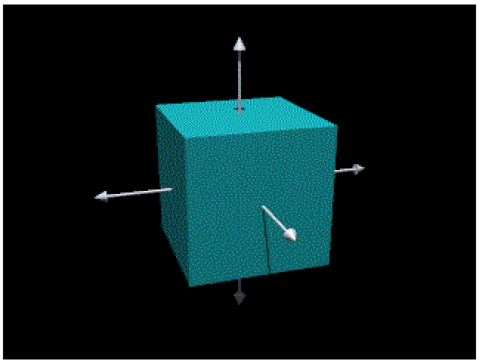
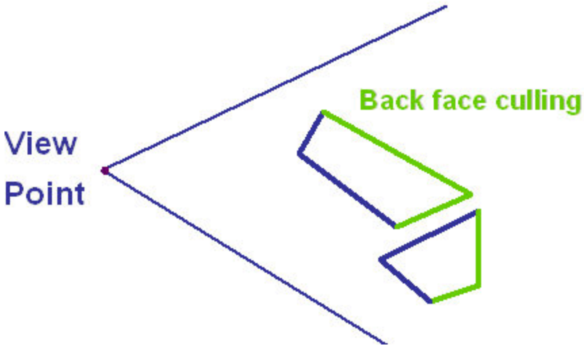
Back-face culling







Back-face culling determines whether a polygon of a graphical object is visible. It is a step in the graphical pipeline that tests whether the points in the polygon appear in clockwise or counter-clockwise order when projected onto the screen. If the user has specified that front-facing polygons have a clockwise winding, if the polygon projected on the screen has a counter-clockwise winding it has been rotated to face away from the camera and will not be drawn.

(http://en.wikipedia.org/wiki/Back-face_culling)

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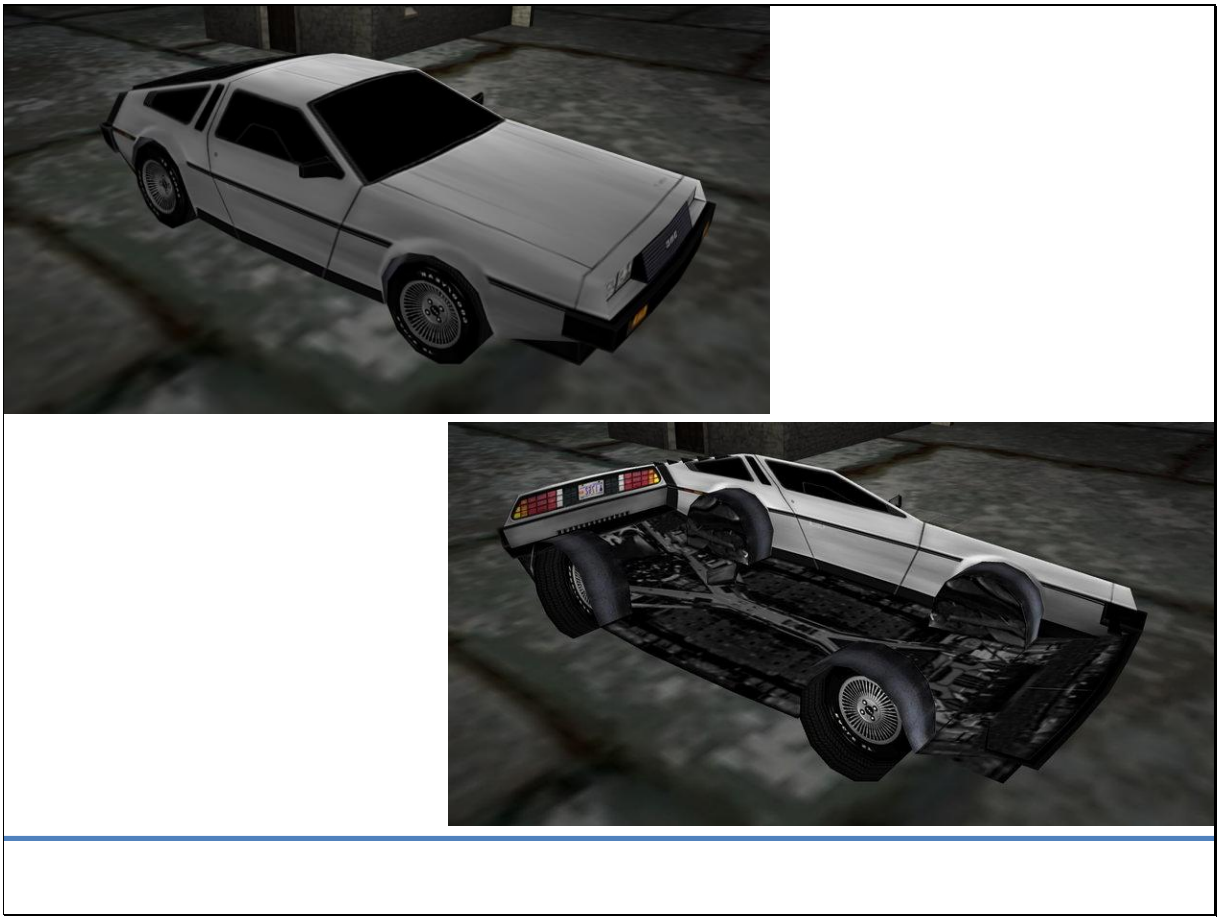

On the slide the same model can be see. The difference is that the determines the parts of the object visible. The first one is right and the second is wrong.

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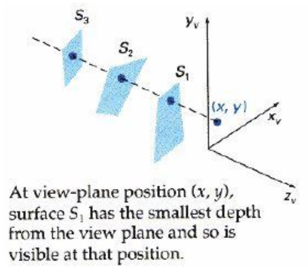
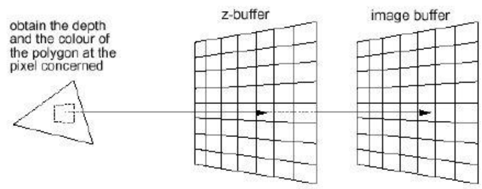
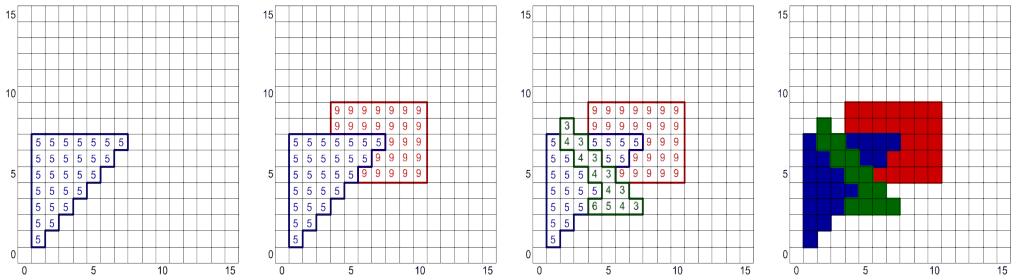
Z-buffer algorithm







Z-buffering is the management of image depth coordinates in 3D graphics. It's usually done in hardware, sometimes in software. It is one solution to the visibility problem, which is the problem of deciding which elements of a scene are visible, and which are hidden.
When an object is rendered by a 3D graphics card, the depth of a generated pixel is stored in a buffer, the z-buffer. This buffer is usually arranged as a 2D array with one element for each screen pixel. If another object of the scene must be rendered in the same pixel, the graphics card compares the two depths and chooses the one closer to the observer. The chosen depth is then saved to the z-buffer and replacing the old one. In the end, the z-buffer will allow the graphics card to correctly reproduce the usual depth perception. A close object hides a farther one.

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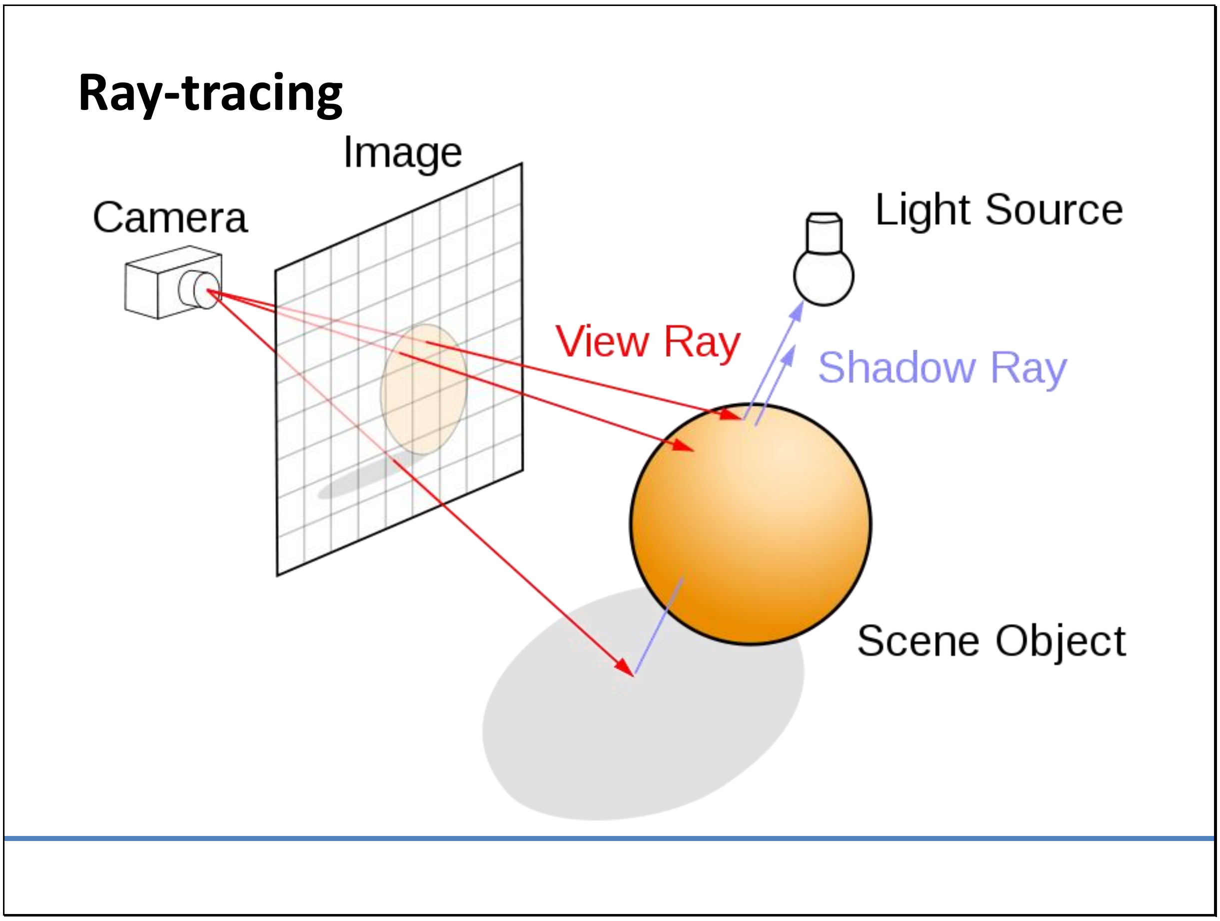

In computer graphics, ray tracing is a technique for generating an image by tracing the path of light through pixels in an image plane and simulating the effects of its sections with virtual objects. The technique is capable of producing a very high degree of visual realism, usually higher than that of typical scan line rendering methods. This makes ray tracing best suited for applications where the image can be rendered slowly ahead of time, such as in still images and film and television special effects. Ray tracing is capable of simulating a wide variety of optical effects, such as reflection and refraction, scattering, and dispersion.
Optical ray tracing describes a method for producing visual images constructed in 3D computer graphics environments, with more photorealism than either ray casting or scan line rendering techniques.

(http://en.wikipedia.org/wiki/Ray_tracing_(graphics))

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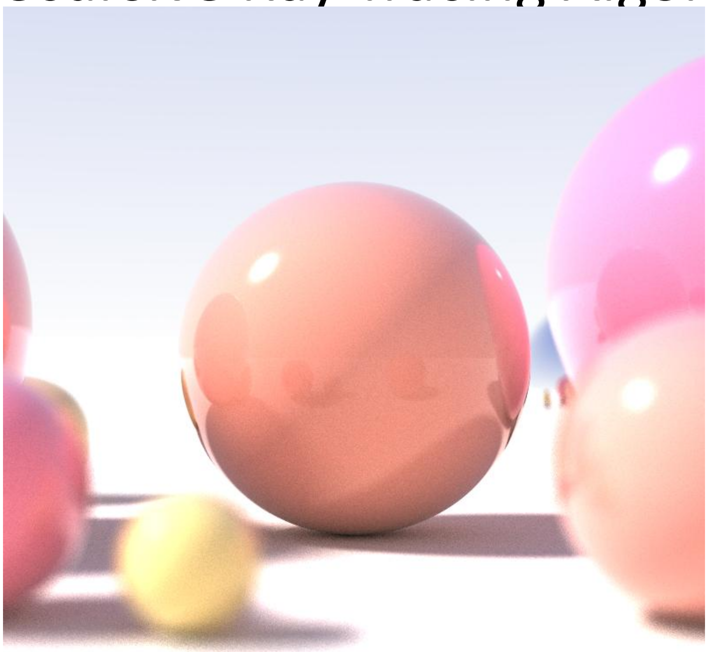
The Recursive Ray Tracing Algorithm



Ray tracing follows reflected and refracted rays through a scene. The rays are thin, so aliasing is a problem. Ray tracing can be used as a basic technique for volume rendering. It's a recursive algorithm. It use secondary rays which are followed recursively from primary rays.

(http://cs.fit.edu/~wds/classes/adv-graphics/raytrace/raytrace.html
Pic.: http://en.wikipedia.org/wiki/File:Recursive_raytrace_of_a_sphere.png)

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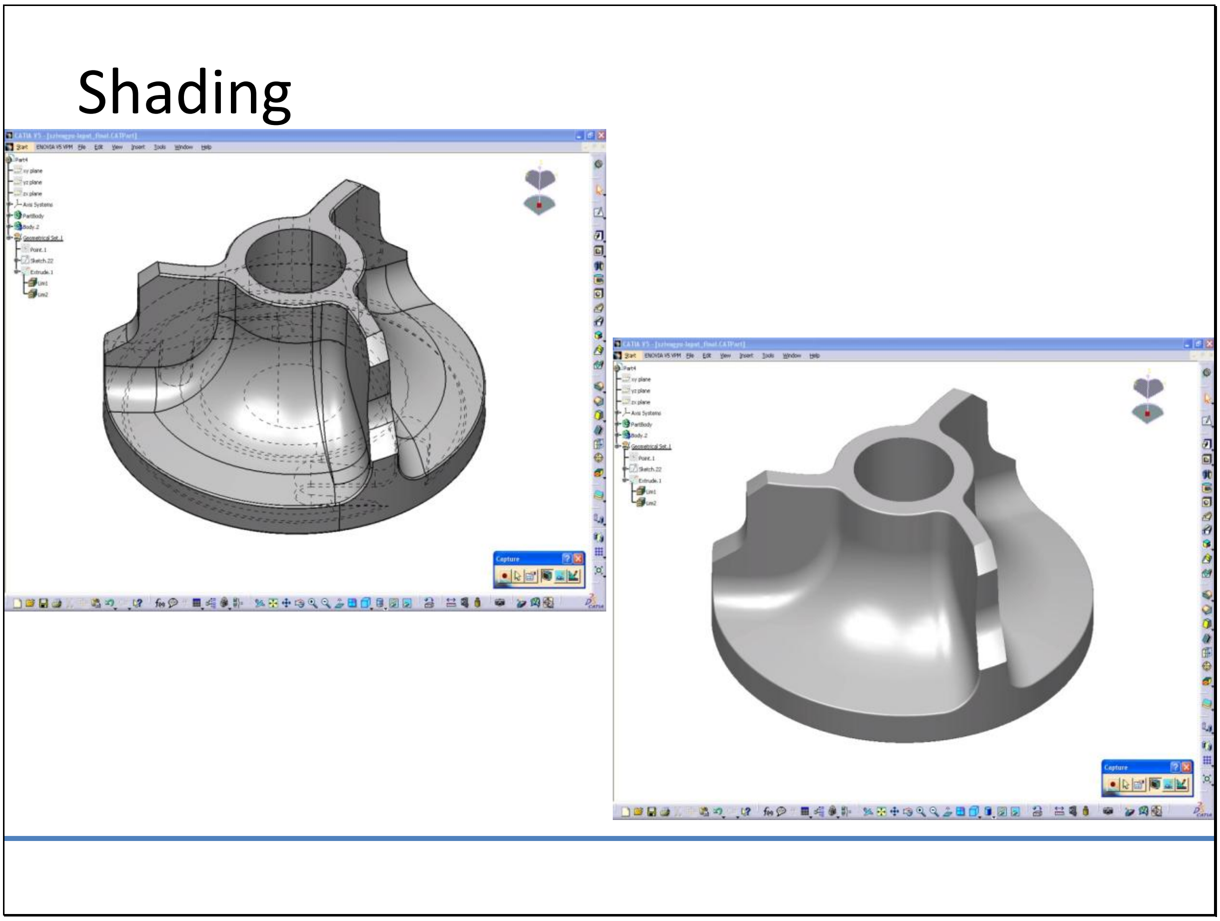

Shading refers to representing depth perception in 3D models or illustrations by levels of darkness, and the process of altering a colour based on its angle to lights and its distance from lights to create a photorealistic effect. Shading is a part of the rendering process.
Shading alters the colours of faces in a 3D model based on the angle of the surface to a light source or light sources.
The first image below has the faces of the box rendered, but all in the same colour. Edge lines have been rendered here as well which makes the image easier to see.
The second image is the same model rendered without edge lines.

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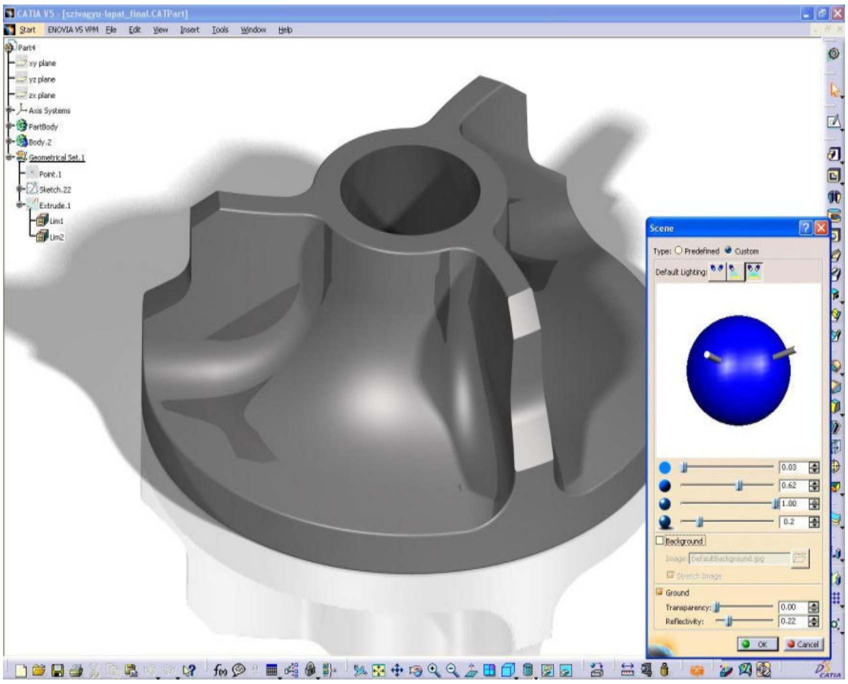
Lights



Lighting fixtures come in a wide variety of styles for functions. The most important functions are as a holder for the light source, to provide directed light and to avoid visual glare. Some are plain and functional and some are pieces of art in themselves.

There are many types of lights can be used. Spotlight, directional or point light for example.

(http://en.wikipedia.org/wiki/Lighting)

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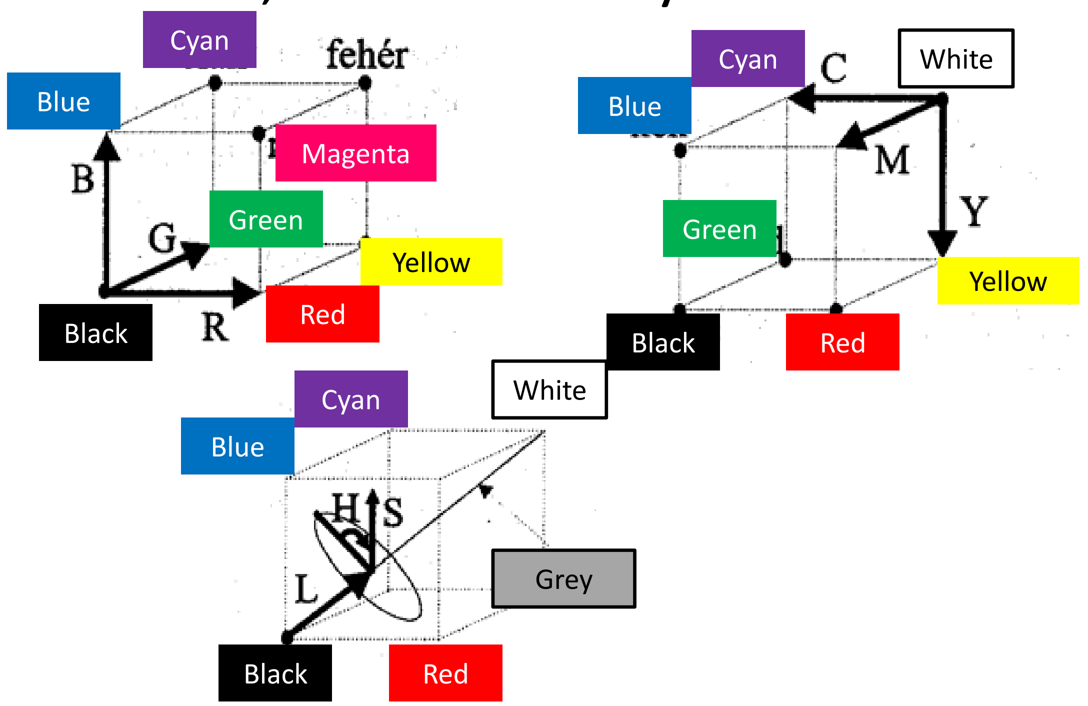
Colours, colour code systems



RGB (Red, Green, Blue)
The RGB colour model is an additive colour model in which red, green, and blue light is added together in various ways to reproduce a broad array of colours. The name of the model comes from the initials of the three additive primary colours, red, green, and blue. [1]

CMY (cyan (C), magenta (M) , and yellow (Y))
It is possible to realize a large range of colours seen by combining cyan, magenta, and yellow transparent inks. These are the subtractive primary colours. Often a fourth black is added to improve reproduction of some dark colours. [2]

HLS
HSL is one of the most common cylindrical-coordinate representations of points in an RGB colour model, which arrange the geometry of RGB in an attempt to be more intuitive and perceptually relevant than the cube representation.
HSL stands for hue, saturation, and lightness. It is often called HLS. [2]

[1] http://en.wikipedia.org/wiki/RGB_color_model
[2] http://en.wikipedia.org/wiki/Color_model#HSV_and_HSL_representations

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Textures

In computer graphics, texture filtering or texture smoothing is the method used to determine the texture colour for a texture mapped pixel, using the colours of nearby pixels of the texture. Mathematically, texture filtering is a type of anti-aliasing, but it filters out high frequencies from the texture fill. It allows a texture to be applied at many different shapes, sizes and angles while minimizing blurriness, shimmering and blocking.

(http://en.wikipedia.org/wiki/Texture_filtering)

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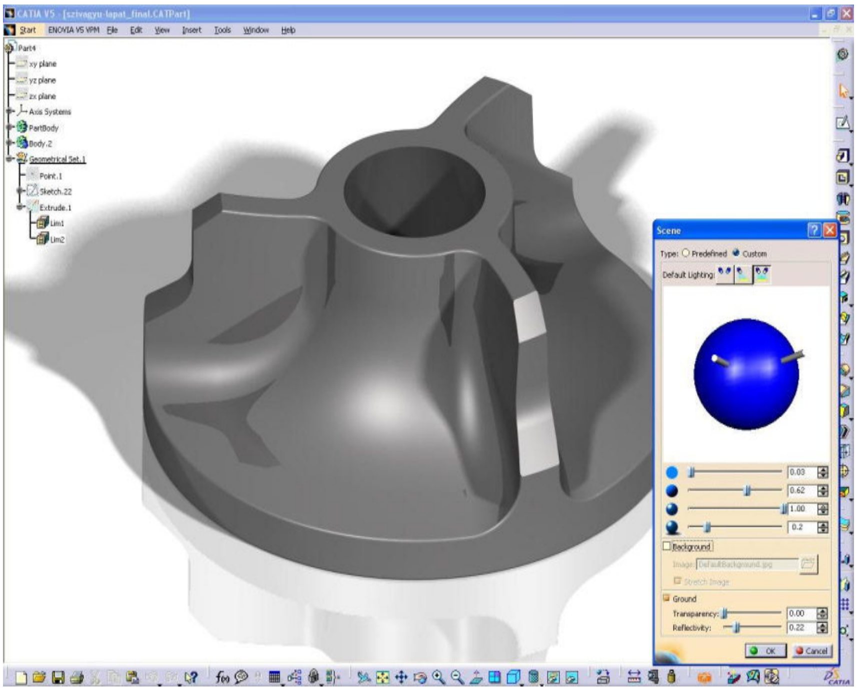
Rendered picture



Rendering is the process of generating an image from a model. A scene file contains objects in a strictly defined language or data structure; it would contain geometry, viewpoint, texture, lighting, and shading information as a description of the virtual scene. The data contained in the scene file is then passed to a rendering program to be processed and output to a digital image or raster graphics image file.

(http://en.wikipedia.org/wiki/Rendering_(computer_graphics))

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CAD Book

3. Geometric modelling

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Introduction

Generally speaking, a model is nothing but the copy of a real or imagined object, mapping thereof using limited information. **A computerized geometric model maps up the shape and dimensions of an object.**

As a result of attempts to develop an ideal geometric modelling system, there is a broad range of methods available today. Nevertheless, no universal solution has been managed to be developed to satisfy all demands for a geometric product model in itself. Known methods offer different application options depending on product and task.

Experience shows that an appropriate in-depth familiarization with the theoretical basics of geometric modelling systems enhances effective modelling work, on the one hand, and accelerates the mastering of CAD systems not used earlier.

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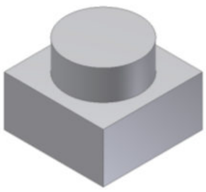
Introduction

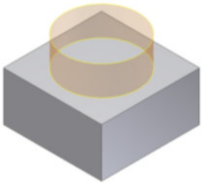
From the topological point of view, geometric modelling systems can be classified into two basic groups:

- **manifold** modelling systems: they include modelling systems suitable for modelling forms that can be mapped into a manifold of 2D points.

- objects of a **non-manifold** topology are not realistic in general; they cannot be mapped into a manifold of 2D points. This usually arises from the fact that a model includes basic units of different dimensions (1D, 2D or 3D) or the latter are interconnected within a model.



manifold



non-manifold

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Manifold modelling systems

Manifold modelling systems can be broken down into two further groups based on the completeness of information on features of shape:

- **Modelling systems of other than full value include:**
  - wireframe modelling
  - surface modelling
- **Modelling systems of full value include:**
  - mantle modelling
  - solid modelling

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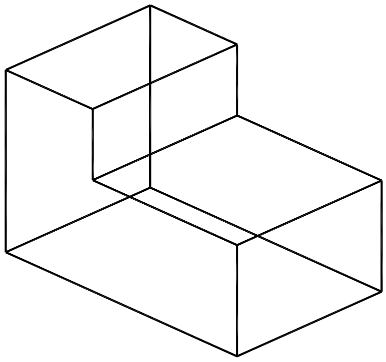
Wireframe modelling

A wireframe model depicts the edges delimiting the surfaces of the object modelled. These edges can consist of lines, arcs, and curves.



Disadvantages of the modelling method:

- all edges are shown on the image displayed; visibility cannot be depicted;
- volume and mass characteristics cannot be specified;
- data provision is lengthy and difficult;
- not suitable for designing shapes and specifying more complex forms.

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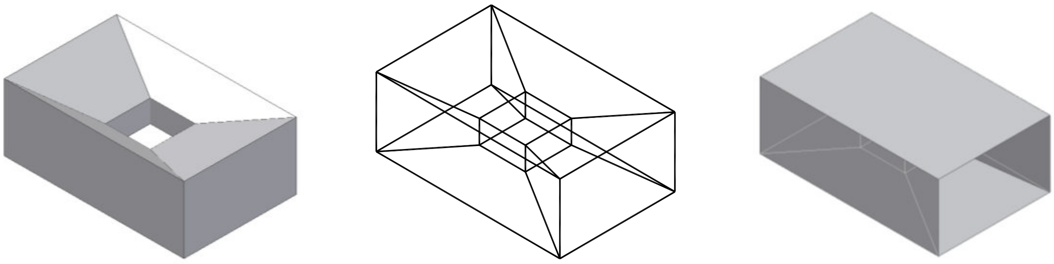
Wireframe modelling

A basic shortcoming of wireframe modelling is that the model displayed does not clearly show the object modelled.



Wireframe modelling is practically out of use today. However,

- in many cases, a wireframe image can be advantageous for model design in mantle and solid modelling;
- wireframe models are built as supporting frames for surface modelling.

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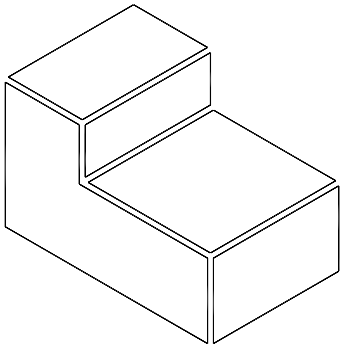
Surface modelling

Surface modelling is aimed at the design of finite, non-open surface patches of free forms, out of which the delimiting surfaces of an object are generated by the geometric positioning of surface patches and by the stipulation of various continuity restrictions. This modelling method does not manage topological information. The non-contacting surfaces on the surface model shown in the figure below are intended to illustrate that surfaces are interconnected only at ”sight” level.



Characteristics of surface modelling:

- a surface model is suitable for hide and shade displays;
- not suitable for calculating volume or mass characteristics;
- not suitable for producing numerical models for engineering calculations.

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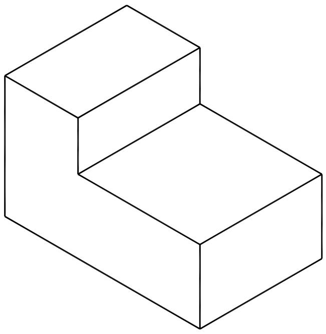
Mantle modelling

Mantle modelling describes the finite and closed cover of an object (the mantle) by a polyeder approach or realistic geometry. Mantle modelling makes methodological use of the basic assumption that each physical object has an unambiguously determinable delimiting surface. Geometrically, this delimiting surface is the mantle, which is a continuous closing set of surface patches. In addition to other information, this modelling method also provides a comprehensive topological characterization of the model.



Characteristics of surface modelling:

- a mantle model is suitable for hide and shade displays;
- is suitable for calculating volume or mass characteristics;
- is suitable for completing production technology designs.

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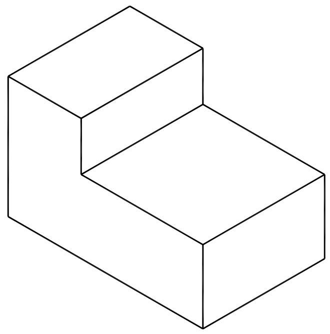
Solid modelling

Solid or volume modelling describes objects as finite, closed, regular sets of points. A solid model provides a complete, characteristic and concise description of the object. The data structure includes descriptions of the basic units building up the solid and their relationships as well. solid modelling is much more simple than wireframe, surface, or mantle modelling.



Many varieties of solid-oriented modelling systems have been developed:

- volume breakdown methods:
  - prism breakdown modelling;
  - half space modelling;
- volume fillup methods:
  - modelling by elemental cells;
  - modelling by elemental solids.

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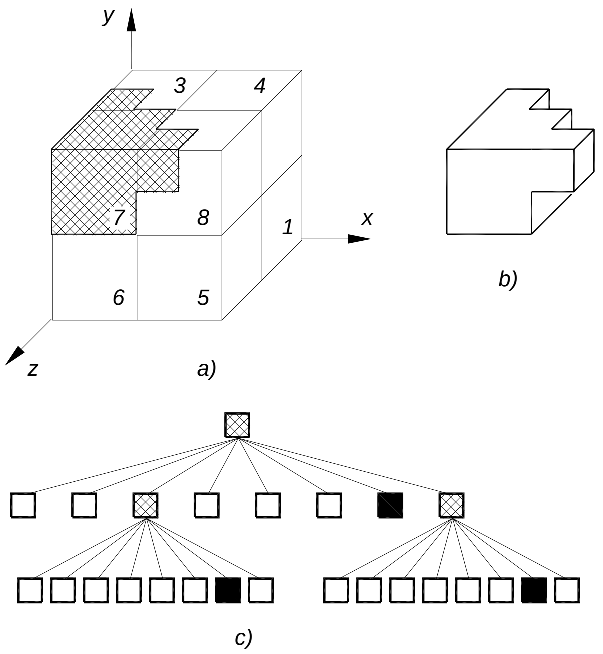
Solid modelling – Prism breakdown method



Modelling based on prism breakdown divides a finite space region into eight parts (producing octants), and then it examines each space region whether they are fully or partly filled up. Partial regions totally filled up or not filled at all can be excluded from further investigations.

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Solid modelling – Prism breakdown method

The octants resulting from the next breakdown of partially filled regions constitute the third level of the hierarchic tree where the procedure described earlier must be repeated.

This method, applying a so-called hierarchical decomposition method, can provide accurate descriptions in case of objects delimited by perpendicular plane surfaces, and only approximate descriptions in case of oblique and curved surfaces. The accuracy of approximation can be influenced by breakdown depth.

Advantages of the procedure include the fact that it is extremely simple to express by algorithms and that its application does not require special user skills.

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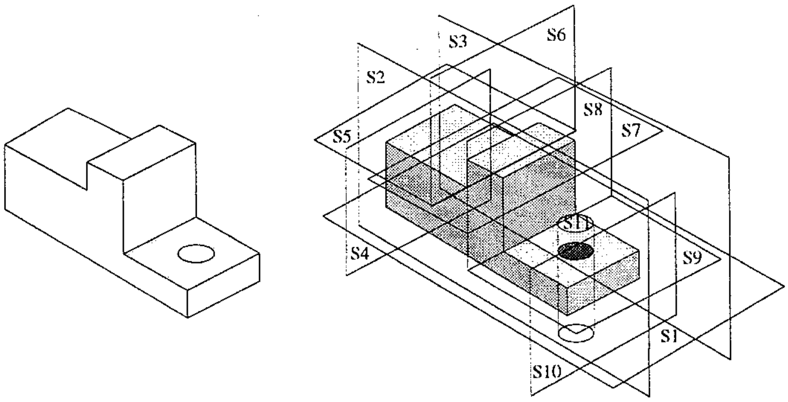
Solid modelling – Half space method

Half space breakdown modelling is characterized by the fact that the volume occupied by the object is delimited by surfaces of infinite extension which divide space into two regions of infinite extension. Such surfaces of infinite extension are laid on the surfaces of the object to be modelled, assuming that the half space on one side of the surface is empty, and the other one is filled by a material.



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Solid modelling – Half space method

Mathematical definition of a half space:

$$H = \{P : P \in E^3 \text{ és } f(P) < 0\}.$$

meaning that point P is a point of half space E^3 if the condition $f(P) < 0$ is fulfilled for the surface equation $f(P)$.

Some examples for surface equations formulated in an implicit form:

Síkfelület:	$\{(x, y, z): z = 0\}.$
Hengerfelület:	$\{(x, y, z): x^2 + y^2 = R2\}.$
Kúpfelület:	$\{(x, y, z): x^2 + y^2 = [(R/H)z]^2\}.$
Gömbfelület:	$\{(x, y, z): x2 + y2 + z2 = R2\}.$
Tórusz:	$\{(x, y, z): (x^2 + y^2 + z^2 - R_2^2 - R_1^2)^2 = 4R_2^2(R_1^2 - z^2)\}.$

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Solid modelling – Half space method

The volume of the solid S is the intersection (common portion) of half spaces H_i.

$$S = \bigcap\left(\sum_{i=1}^{n} H_i\right)$$

For instance, a rectangle can be described as the intersection of 6 half spaces.

A disadvantage of half space modelling is that users must be highly familiar with modelling-related laws, because otherwise a non-closed object can easily be generated.

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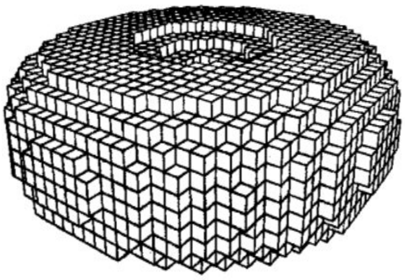
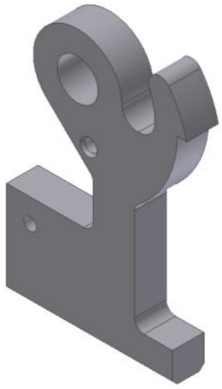
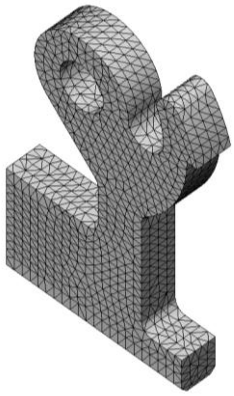
Solid modelling – Modelling by elemental cells

In modelling by elemental cells, components are built up by so-called **isomorphic cells**, which are smaller by several orders of magnitude than the component size. Modelling by elemental cells primarily serves as a modelling tool for numerical procedures (finite element, boundary element methods). The figures below show modelling by elemental cells, the 3D geometric model of a component, and its finite element model made up of small tetrahedron elements.







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Solid modelling – Modelling by elemental solids

In modelling by elemental solids, components are built up of so-called **solid primitives**, falling into the order of magnitude of their dimensions and having a specific geometry, by using composition operations.

The English term for the modelling procedure to combine elemental solids is:

Constructive Solid Geometry

or **CSG** modelling for short.

Modelling by elemental solids is the most widespread of all volumetric modelling methods. Later on, this modelling form will be termed as **solid modelling**.

A **solid model** provides a complete, characteristic and concise description of an object, enabling integrated and automated design.

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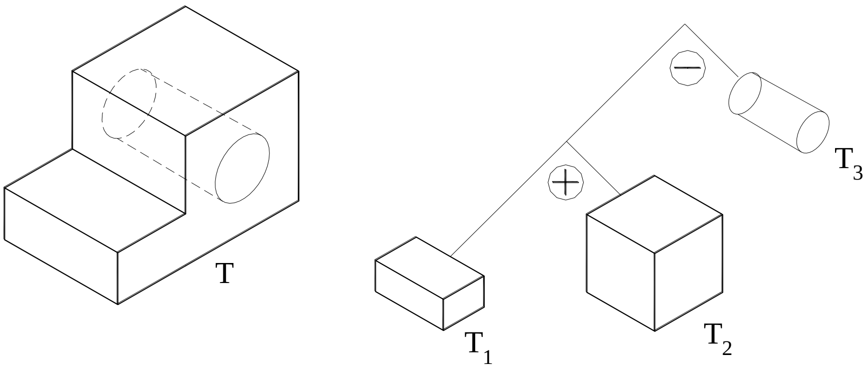
Solid modelling – Modelling by elemental solids

The two basic toolset groups of solid modelling are elemental geometric solids T_i and **composition operations** ⊗ (⊗ is a summary indication of composition (set) operations).

The complex solid **T** as in the figure is generated by adding primitives $\mathbf{T}_1$ and $\mathbf{T}_2$ and subtracting primitive T_3.



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Assumptions in solid modelling

- an object is a rigid solid, meaning that it has a concrete and invariant shape not affected by spatial location or position;
- an object fills the space occupied by it homogeneously, meaning that the inside of the object is always connected to the complementary of the model through the cover;
- object extension is finite, meaning that the model can be mapped for computer display;
- an object can be generated as a composition of a finite number of elemental solids, meaning that an object model can be stored in a computer;
- an object can be modelled as a closed set in terms of rigid solid motion.

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Solid modelling – approach by the theory of sets

Let the space regions occupied by elemental geometric solids (solid primitives) be:

$$T_1, T_2, T_3 \ldots T_i \ldots T_n$$

A complex solid, that is, an object, can be generated by the composition of elemental geometric solids:

$$T = \otimes(T_i) \qquad 1 \leq i \leq n$$

where ⊗ indicates possible composition operations:

$\cup$	union;
$\setminus$	difference;
$\cap$	intersection.

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Solid modelling – approach by the theory of sets

The previous equation developed:

$$T = ((((T_1) \otimes T_2) \otimes T_3) \ . \ . \ .)$$

If the regions T_i are regular, the equation above produces a mathematically complete and unique result object, but the composition (the method of generation) is not unambiguous. This means that the same result object can be generated from other solid primitives T_i and by other composition operations as well.

Geometrically, the size of elemental geometric solids T_i is similar to the order of magnitude of the model T and their number is finite.

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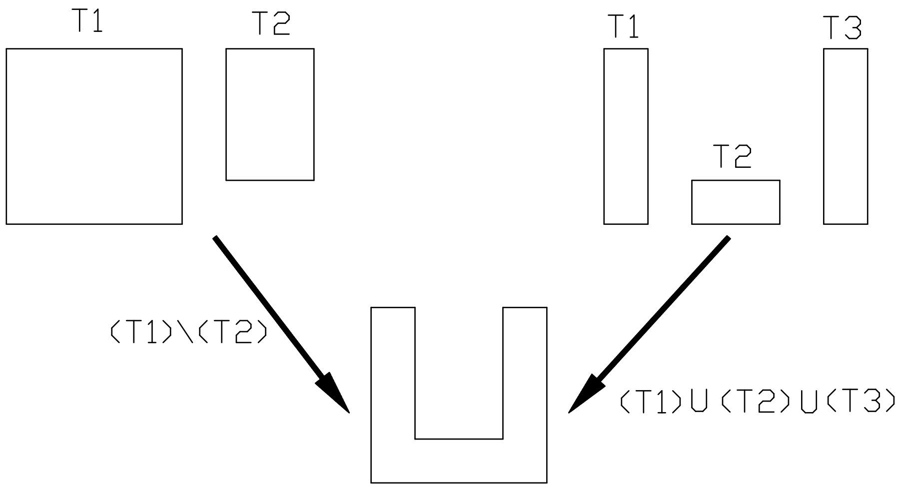
Solid modelling – approach by the theory of sets



If the regions Ti are *regular*,
the result object will be *complete* and *unique*.

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Solid modelling – approach by the theory of sets

The solid model T is defined as a point set in space. The limits of the object divide space into an external and internal point set. By introducing the following signage:

- bT is the inside of the model;
- hT is the boundary of the model;
- kT is the complementary set of points of the model (that is, the points outside of it).

The complete model space can be stated as follows:

$$M = \mathrm{b}T \cup \mathrm{k}T \cup \mathrm{h}T$$

The model itself, representing the inside of the model and the boundary of the model:

$$T = \mathrm{b}T \cup \mathrm{h}T = \mathrm{bh}T$$

The mantle model: hT

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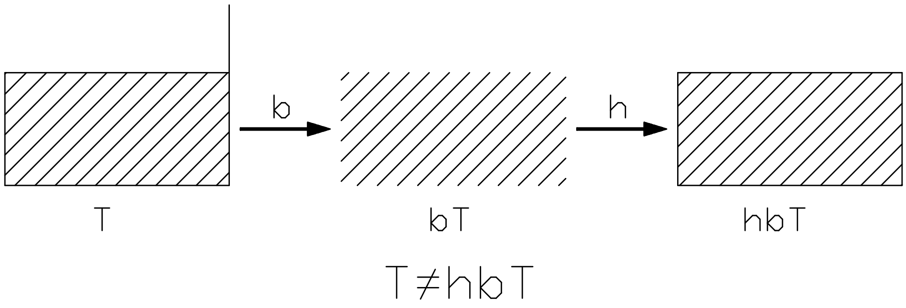
Solid modelling – approach by the theory of sets

Regions T_i must be closed and regular. A region T is regular if the following condition is met:

$$T = \mathrm{hb}T$$



Example of a non-regular region.

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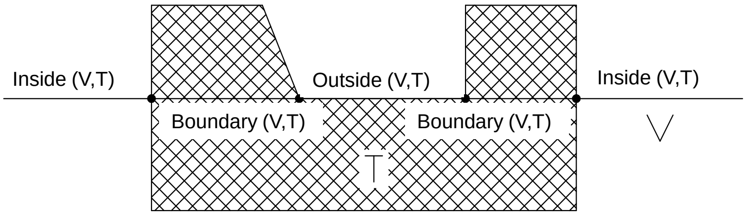
Solid modelling – approach by the theory of sets

A special area of solid modelling, not at all unproblematic, is to find out whether certain points are included in a given region. Inclusion information is important for

- surface-like display;
- engineering quantity calculations;
- crash tests.

The following example shows three relations between the points of a set *V* intersecting a regular set *T.*



$P \in bT$ inside;

$P \in hT$ boundary;

$P \in kT$ outside.

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Tools for solid modelling

The toolset for solid modelling includes the following:

- creation of solid primitives;
- composition operations;
- manipulation of solid primitives and solids;
- illustration.

Creation of solid primitives

Elemental solids, also named as **solid primitives** can be pre-defined or created by the user. **Pre-defined solid primitives** include**:**

– rectangle	– wedge;	– cylinder;
– cone;	– torus;	– sphere.

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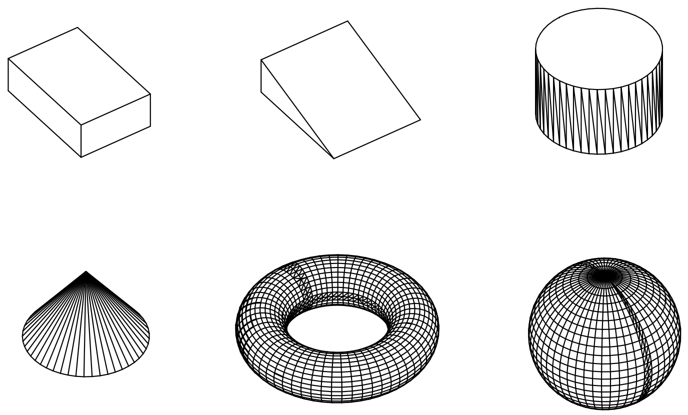
Tools for solid modelling



Pre-defined solid primitives

Some softwares may include additional solid primitives; e.g. cone, convex arc, concave arc, etc.

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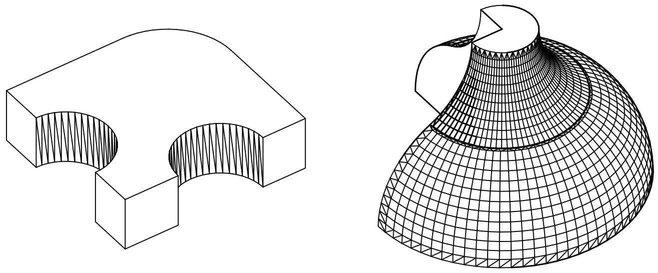
Tools for solid modelling

User-created solid primitives:

– extrude; – revolve; – (sweep); – (loft).



User-created solid primitives

A common feature of user-created solid primitives is that they can be created by moving surfaces.

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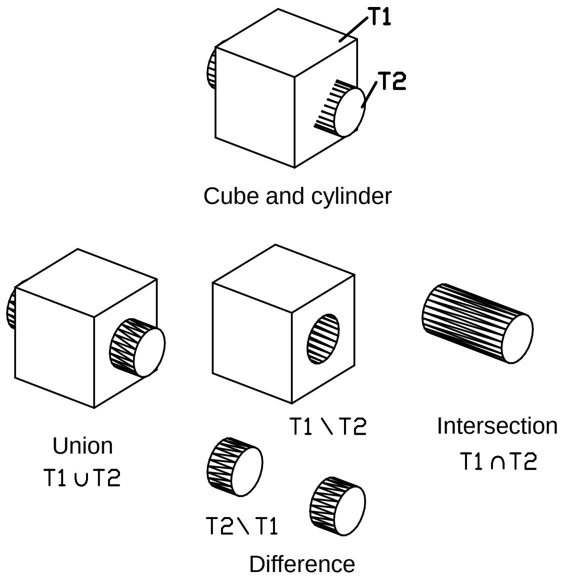
Tools for solid modelling

Composition operations.



Composition operations of two operands include **union** (∪), combining the sets of points of two discrete solids; **difference** (\), producing the difference of two sets of points; and **intersection** (∩), defining the common set of points to be found in both solids.

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Tools for solid modelling

Manipulation of solid primitives and solids

The toolset for solid modelling includes the manipulation of solids and solid primitives, such as:

- move;
- copy;
- revolve;
- mirror;
- scale;
- assign;
- delete;
- etc.

Procedures for illustration

- wireframe;
- hide;
- shade.

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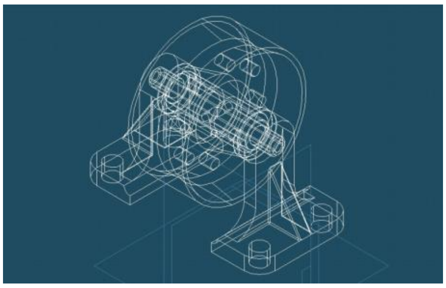
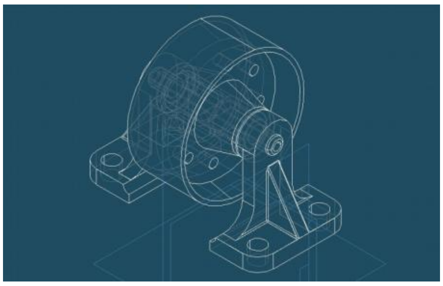
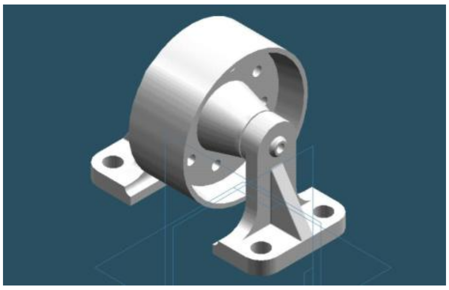
Tools for solid modelling



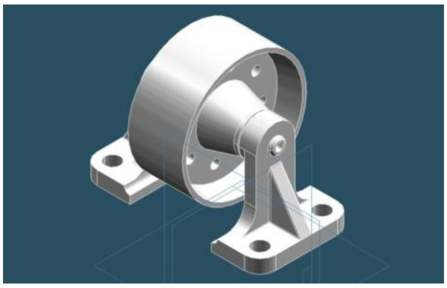
wireframe



hide



shade



shade + hide

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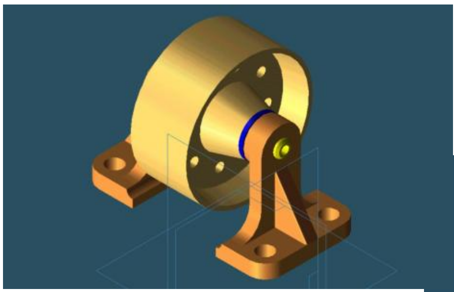
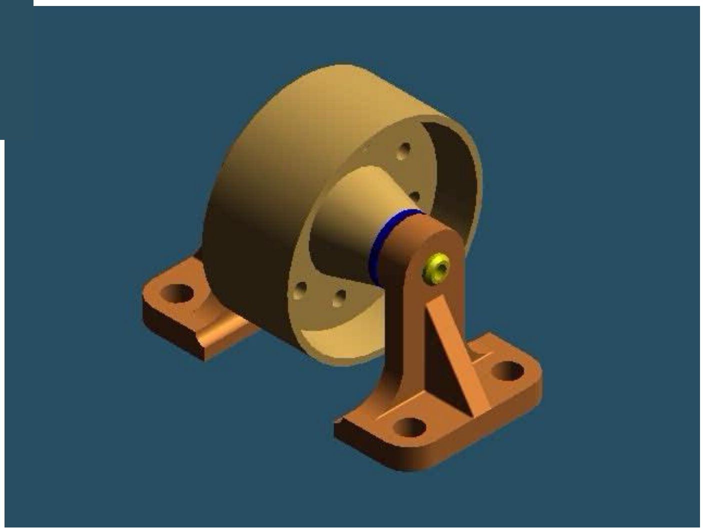
Tools for solid modelling



rotating illustration

with material specifications



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Solid modelling

In practice, the solid modelling process consists of defining solid primitives, adjusting dimensions, transformation into the appropriate position, and the application of generalized set operations. The benefit of combining elemental solids is that it ensures verisimilitude for the model produced.

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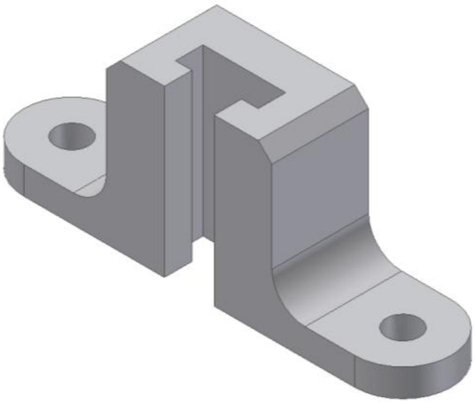
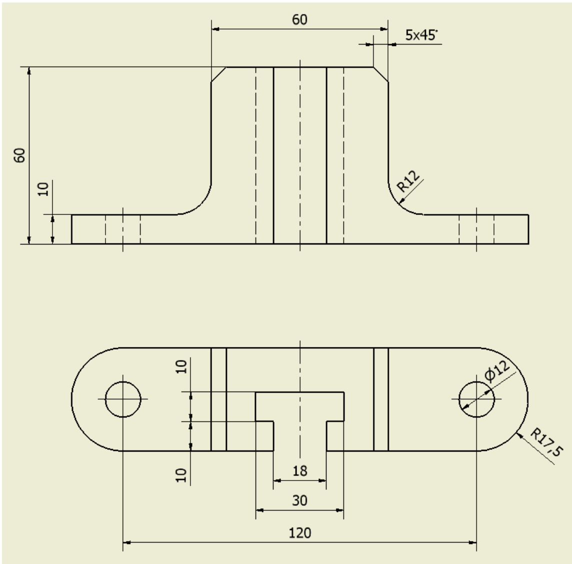
Example for producing a solid model





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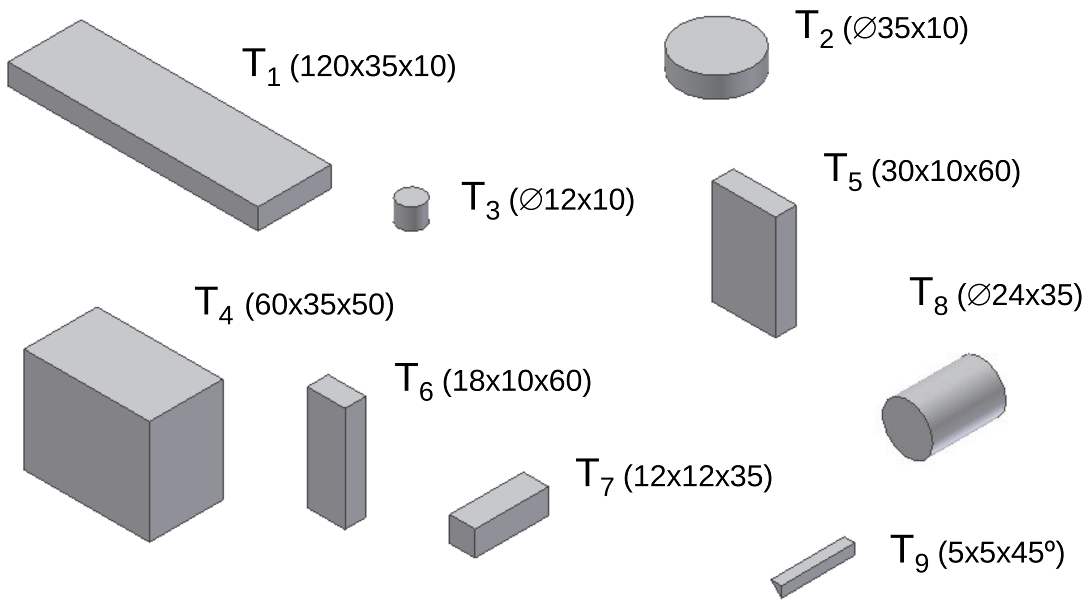
Example for producing a solid model

Solid primitives required for producing a solid:



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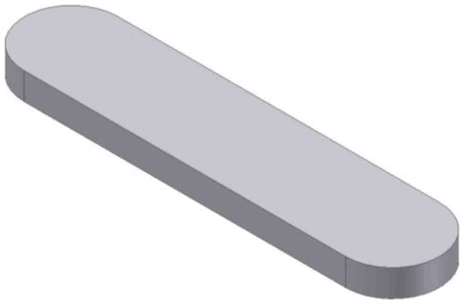
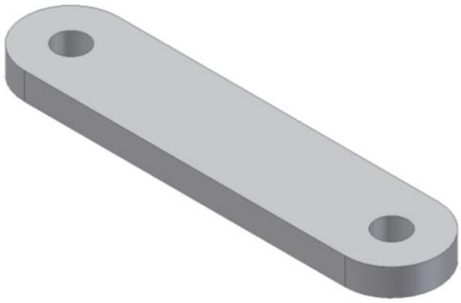
Example for producing a solid model

Model design process (1/4)

$T^1=(((T_1)\cup T_2)\cup T_2)$



$T^2=(((T^1)/T_3)/T_3)$



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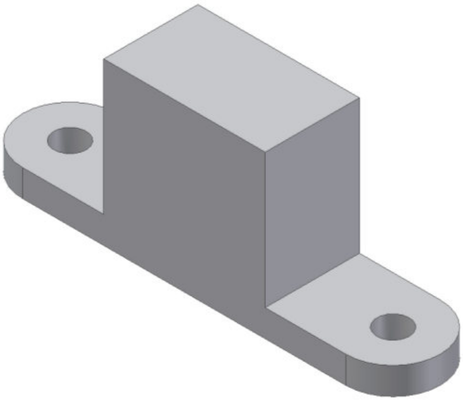
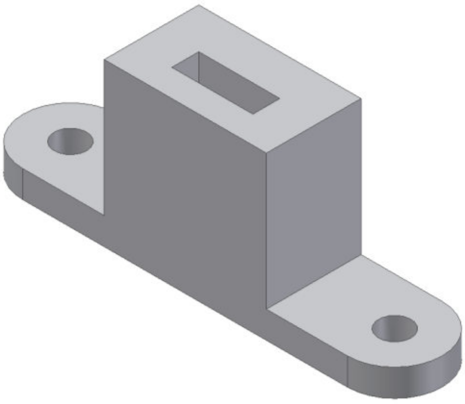
Example for producing a solid model

Model design process (2/4)

$T^3=((T^2)\cup T_4)$



$T^4=((T^3)/T_5)$



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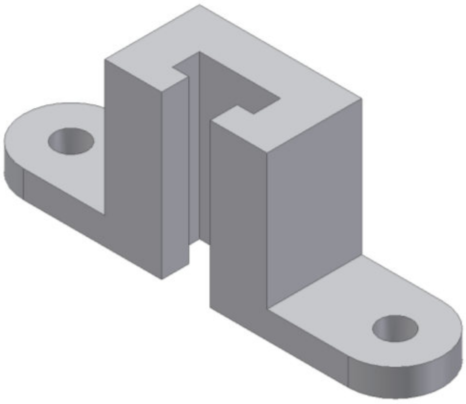
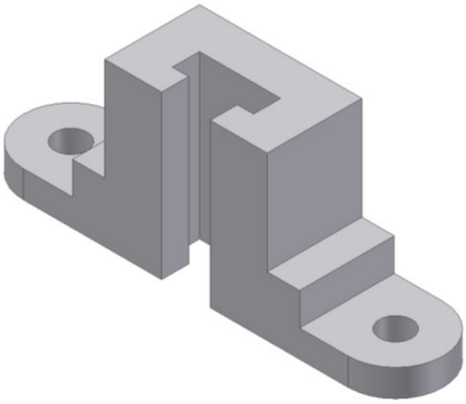
Example for producing a solid model

Model design process (3/4)

$T^5=((T^4)/T_6)$



$T^6=(((T^5)\cup T_7)\cup T_7)$



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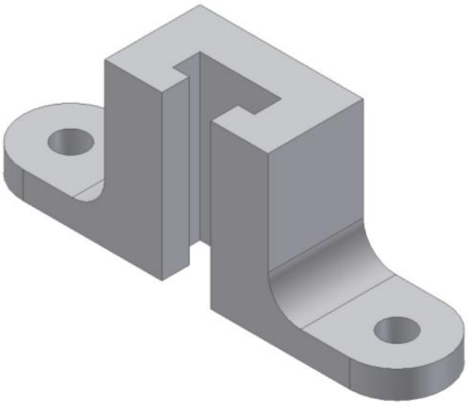
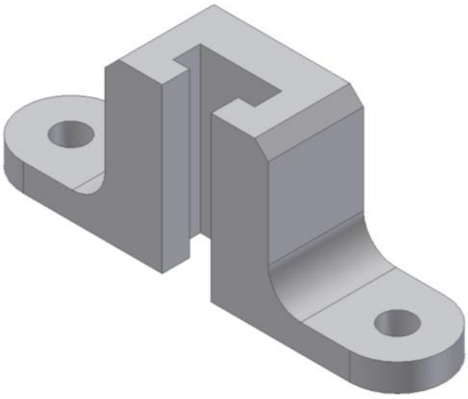
Example for producing a solid model

Model design process (4/4)

$T^7=(((T^6)/T_8)/T_8)$



$T^8=(((T^7)/T_9)/T_9)$



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Limits of solid modelling

By the application of solid modelling, considerable results have been brought about in 3D geometric modelling; however, limitations became evident already in the 1980s; there has been no success in breaking them through to this day. Some of them are as follows:

a) Commercially available modelling systems only provide lower-level basic units for modelling than would be required in engineering practice.

b) Geometric modelling systems do not support engineering thinking, meaning that a final model is produced from a theoretical sketch through continued modifications. Therefore traditional geometric modelling is rather reconstruction than actual design.

c) Geometric modelling systems do not provide a comprehensive description of the object modelled. Thus, they do not provide information on microgeometry, materials, and physical characteristics, all of which are important for operation, manufacturing, control, etc.

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Limits of solid modelling

Elimination of the deficiencies mentioned required the development of systems close to engineering thinking both in terms of content and handling.

In the course of modelling, these systems have to be able to describe **object-related processes** as well in addition to **objects**, so they must manage all the information characterizing the **entire lifetime of the product**.

In order to integrate engineering activities, **product models** should be considered instead of geometric models.

This is made possible by **feature-based design**.

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CAD Book

4. Feature-based geometric modelling

Authors: László Molnár
Dr. Károly Váradi

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Features

The principles of feature-besed modelling were posited by **M. Bunge**.

”The physical world consists of things which are considered to be **objects** regardless of their content. Objects can be characterized by their features known or to be detected by scientific instruments. **Features are quality and quantity characteristics, together with the correlations between them.**”

In terms of design, products and their various parts can be interpreted as objects, while features are characteristics associated with them. Relations between characteristics are described and regulated by correlations and restrictions.

As regards mechanical products, the geometric form is of primary importance in respect of material realization, therefore it seems to be natural that features should be derived from geometry here.

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Features

The characteristics induced by a geometric form are defined as **features of shape**.

Similarly to objects, processes also have quality and quantity characteristics; they are called **features of process**.

Characteristics of the operation of mechanical structures can be summarized as **features of operation**. Natural scientific phenomena serving as a basis for product operation are termed **features of meaning**.

Features can be interpreted according to three approaches:

- geometric interpretation;
- application-oriented interpretation;
- ontologic interpretation.

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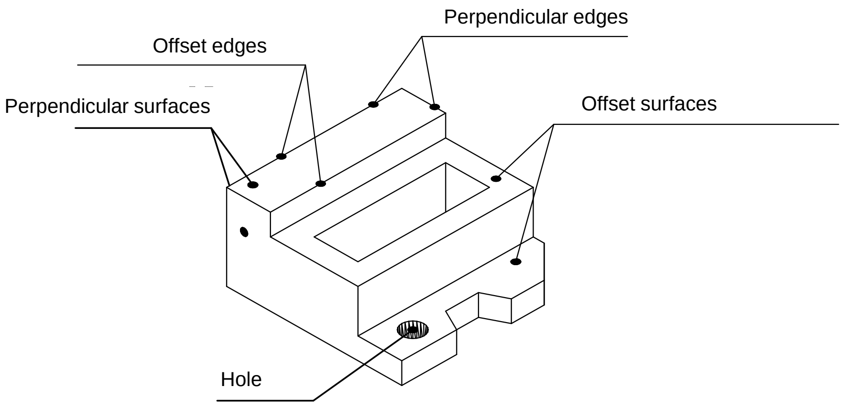
Geometric interpretation of features

According to the geometric interpretation, **features can be considered as a set of information containing the logical association of the points, edges, and surfaces of a component.**



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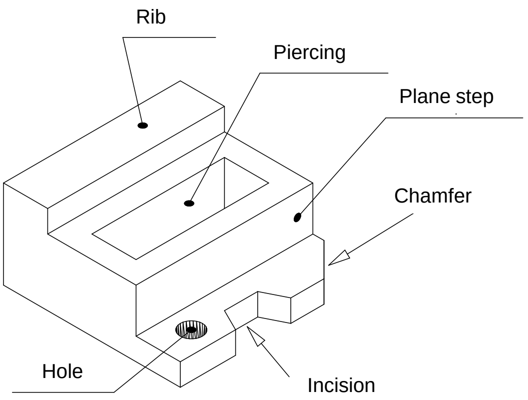
Geometric interpretation of features

Another way of geometric interpretation is to take application concerns more into consideration: **a feature is a basic geometric unit constituting that given area of the shape of the object modelled which is of importance as regards product realization.**



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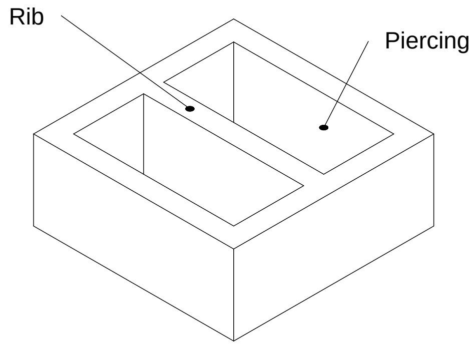
Geometric interpretation of features

A geometric approach to the interpretation of features is problematic because it is not explicit.



For a designer, a rib – as a load-bearing element – is a basic feature. For a technologist, a rib – as a unit to be shaped – is a basic element. If both are integrated in the model, it will become over-defined.

Breakdown of an object into features is not explicit as it depends on the purpose of utilizing the model.

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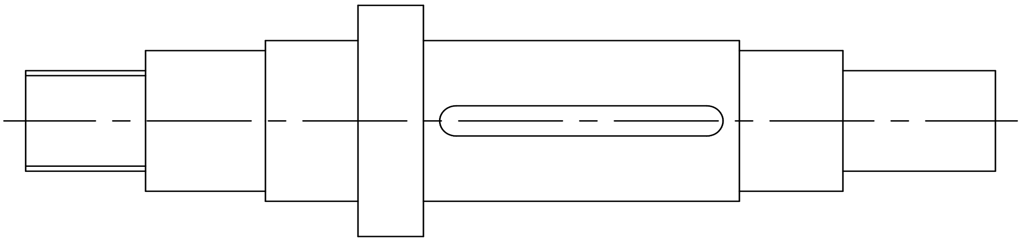
Semantic interpretation of features

More advanced tools for modelling geometric features already enable the **management of attributive information in addition to shape**, constituting the first step towards semantic orientation. According to the semantic interpretation of features, they can be classified into shape-generating, shape-modifying, shape-independent and shape-neutral feature types.

A **shape-generating feature** means a closed form required to perform a function**.** It is also termed as a **carrier form**.



Shape-generating

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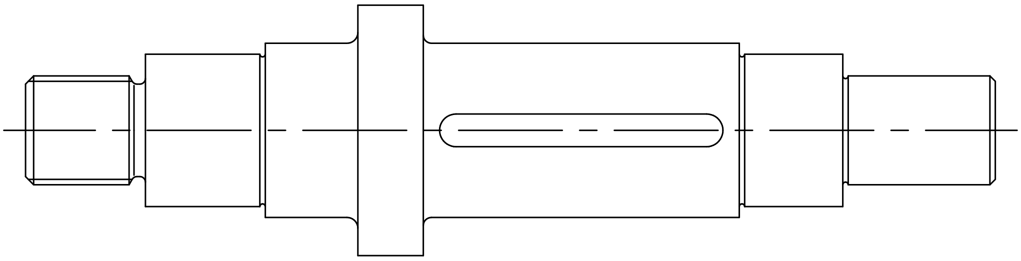
Semantic interpretation of features

Shape-modifying features modify carrier features based on production, installation, strength, etc. considerations.



Shape-modifying

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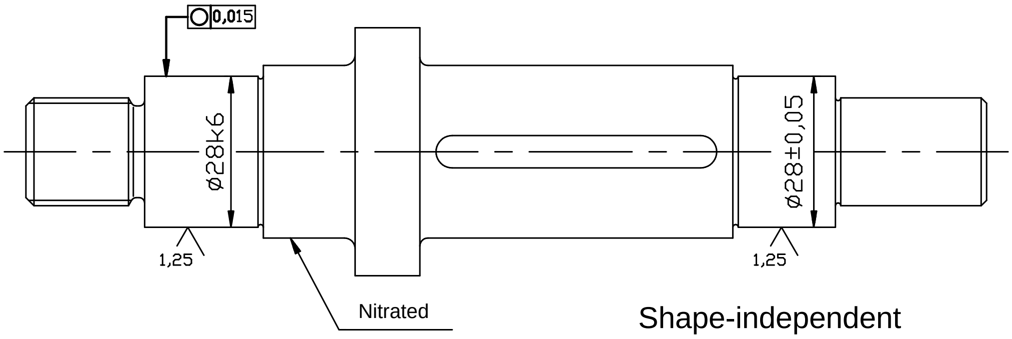
Semantic interpretation of features

Shape-independent features are linked to the nominal shape, causing only its secondary modification, though. This modification is not followed by the geometry; it is only included in the technical specifications. Such shape-independent features include dimension tolerance, surface roughness, surface treatment, etc.



Shape-independent

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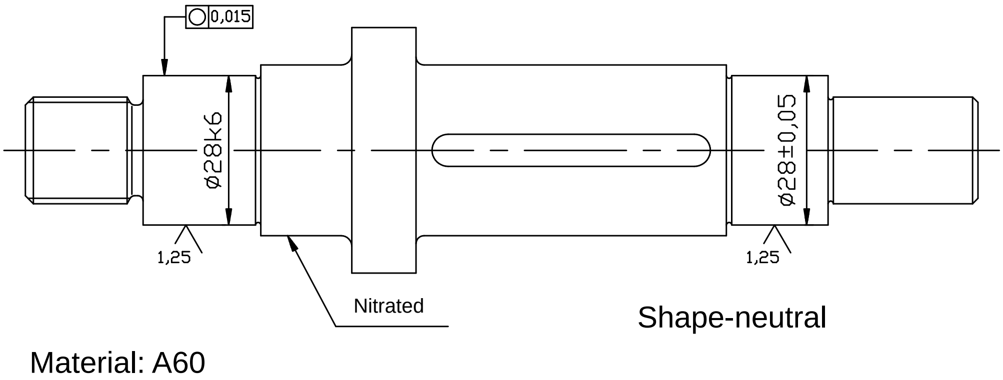
Semantic interpretation of features

Shape-neutral features are not directly related to geometry; they are treated as attributes only. They include material quality, heat treatment requirements, etc.



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Ontologic interpretation of features

The **ontologic** interpretation of features is in the research phase at present. In the ontologic interpretation, features constitute high-level basic units of a **product description language**.

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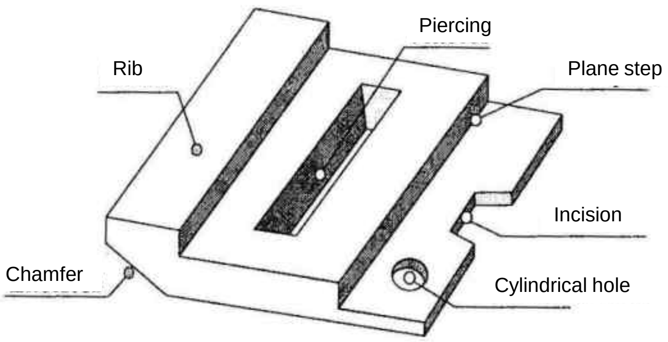
Application-related features

Production technology features

Forms to be shaped and detached by a rotating cutting tool are described by **production technology features**. **Production technology features** can usually be derived directly from **design features**.



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Application-related features

Analysis features

Analysis features are associated with the idealizability of the geometric model used as a basis for strength tests, with the support and load criteria of the model. Accordingly, there are:

- substituting features;
- impact mediating features.

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Application-related features

Installation features

The assembly relations and connection quality of components and parts can be characterized by **installation features**. These include:

- features in a direct connection;

  (these components are in contact or in a specific geometric relation with each other along their surfaces, edges or characteristic points).
- features with an indirect impact;

  (they describe inclusion or a spatial relationship arising from an arrangement structure).
- features describing manageability;

  (they express the possible forms of connection of fixing, installation and support tools).

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Component modelling

In the course of design, the initially conceived shape must be modified several times before reaching the final shape. This is required so because there are function, strength, quality, production, installation, etc. criteria for a form, which can only be implemented and checked separately – or parallelly at best. Today, CAD systems are already required to support the interactive production of design modifications. To our current knowledge, this expectation is met by feature-based softwares when models are determined by geometric and dimensional constraints.

All 3D modelling systems available today are feature-based parametric modelling tools. The most well-known and widespread feature-based parametric design softwares are Mechanical Desktop, Inventor, Solid Works, Solid Edge, Pro Engineer, Catia, NX, and Ideas. Component modelling is a basic module of each system.

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Component modelling

All 3D modelling systems available today are feature-based parametric modelling tools. Component modelling is a basic module of each system.

Main work phases of component modelling:

- produce sketch, providing it with geometric and dimensional constraints;
- create base and further features by adding or removing material;
- modify component as necessary;
- associate material and any other attributive information.

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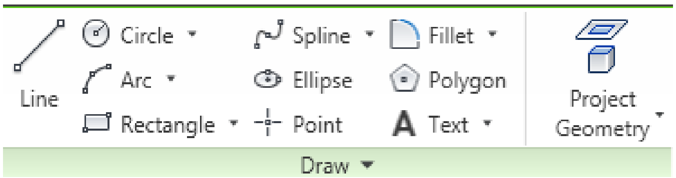
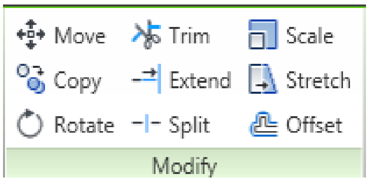
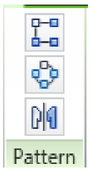
Component modelling - Creating sketches

Sketching is two-dimensional. Drawing and editing commands known from 2D are available for sketching.







The drawing elements of the sketch are linked by geometric constraints. Usual geometric constraints include:

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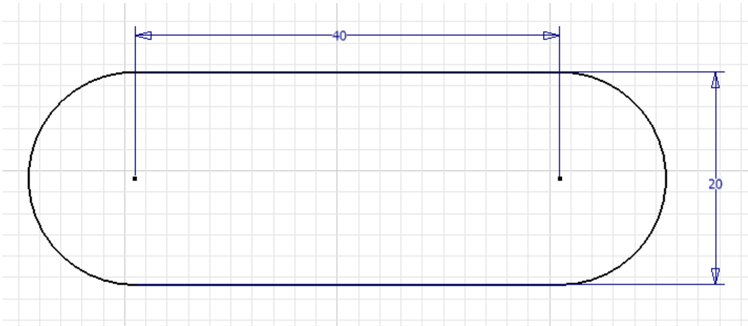
Component modelling - Creating sketches

Example for applying geometric and dimensional constraints on a sketch (1/4)

Task: complete the draft shown in the figure.



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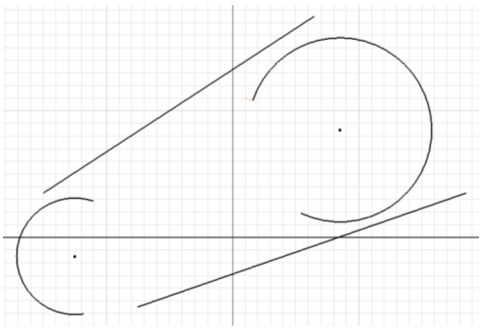
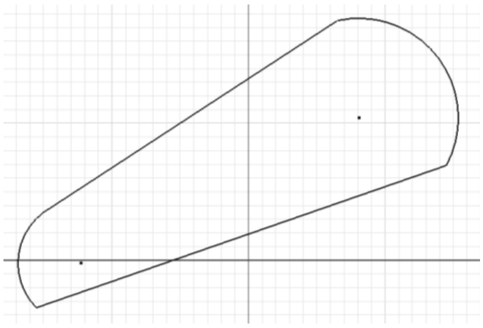
Component modelling – Creating sketches

Example for applying geometric and dimensional constraints on a sketch (2/4)

Use drawing commands to draw the sketch which is at least topologically similar to the form to be created.



Combine the end points of the curves by using the **coincident** constraint.



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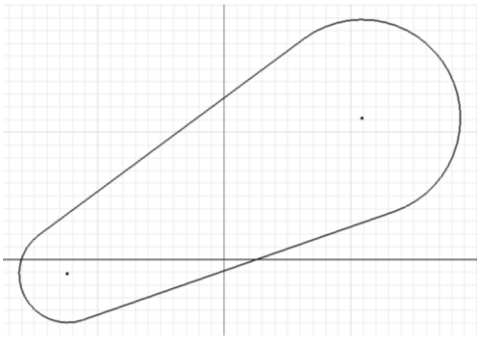
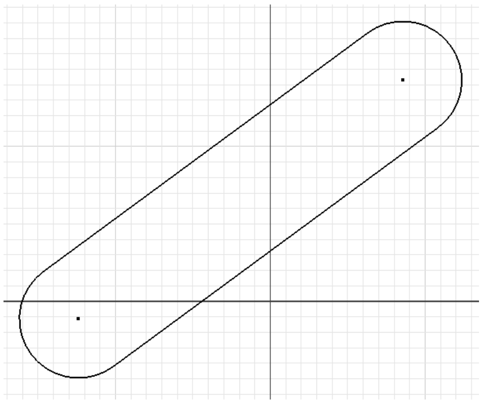
Component modelling – Creating sketches

Example for applying geometric and dimensional constraints on a sketch (3/4)

The **tangent** constraint is used to specify that straight lines and curves touch.



The **offset** constraint can specify that the two straight lines should be parallel.



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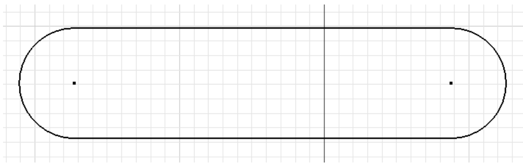
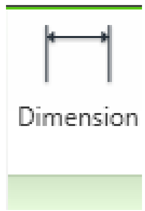
Component modelling – Creating sketches

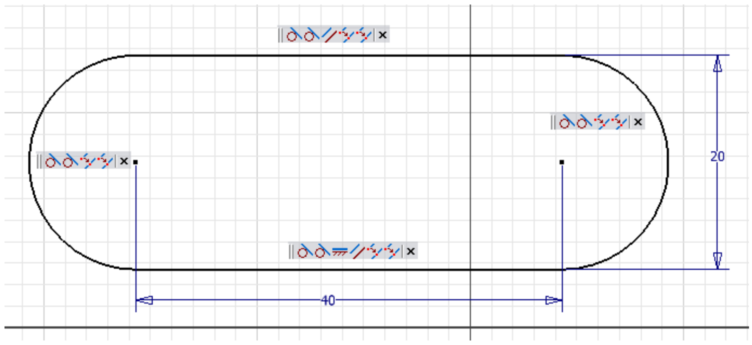
Example for applying geometric and dimensional constraints on a sketch (4/4)

The figure can be made horizontal by a **horizontal** constraint.



A sketch can be defined by dimensional constraints.





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Component modelling – Creating sketches

Notes to the work phase of creating sketches:

- geometric and dimensional constraints can replace and substitute each other;
- softwares do not allow sketches to be over-specified;
- constraints already set can be modified or deleted;
- some softwares assist sketching by automatic constraints.

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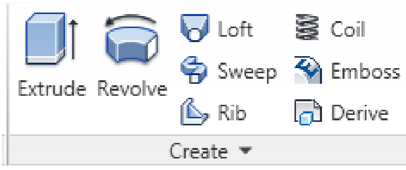
Component modelling – Creating features

Features can be classified into three basic groups:

- sketch-based features;
- located features;
- work features.

Sketch-based features can be generated from previously created sketches. The feature created first – the so-called base feature – can be based on a sketch only.



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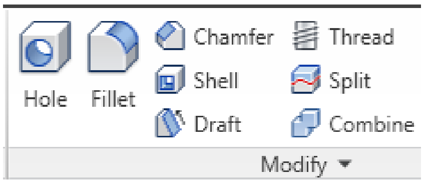
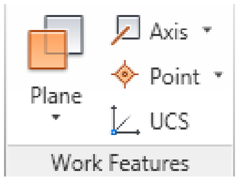
Component modelling – Creating features

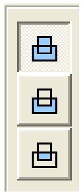
Form elements frequently recurring in design - hole, fillet, chamfer, etc. - can be created without a sketch: they are termed as located features.



Modelling is enhanced by work features.



The **Union**, **Difference** and **Intersection** commands serve for executing set operations between features.



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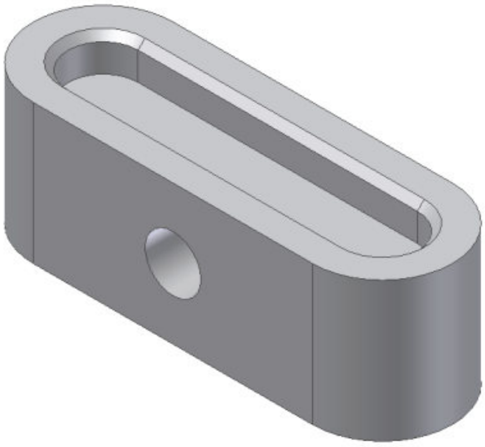
Component modelling – Creating features

Example for creating features (1/4)

Task: produce the component below relying on the sketch created earlier.



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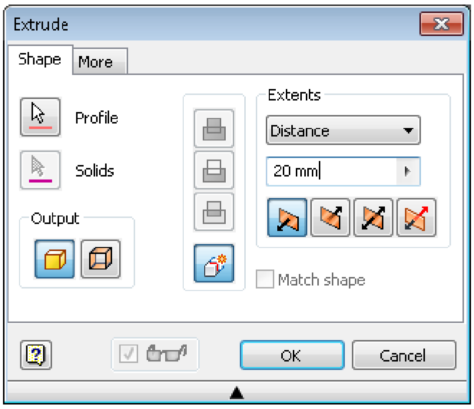
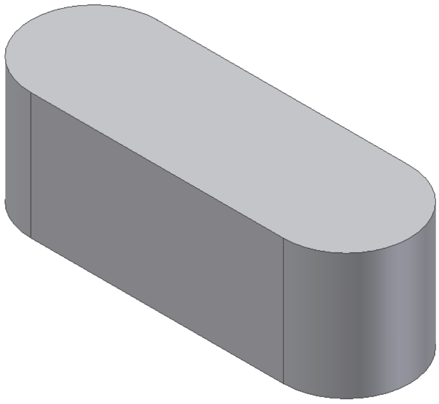
Component modelling – Creating features

Example for creating features (2/4)

Create a basic feature. Extrude the profile 20 mm upwards using the **Extrude** command.





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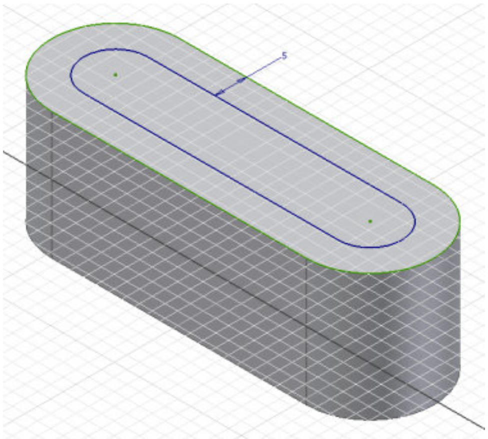
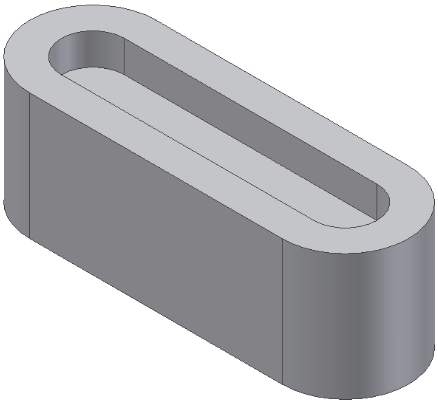
Component modelling – Creating features

Example for creating features (3/4)

Make a recess sketch.



Extrude the profile in difference mode.



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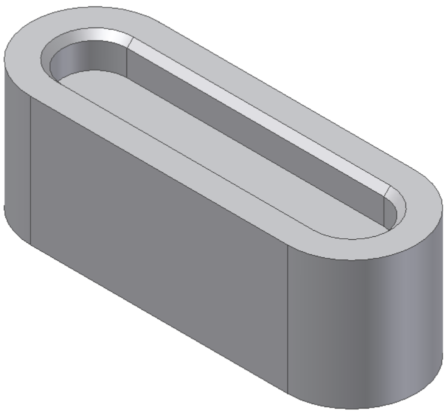
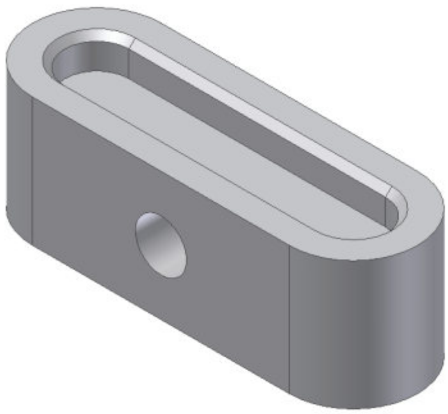
Component modelling – Creating features

Example for creating features (4/4)

Create a chamfer by 1x45°as a located feature.



Create a ∅8 hole as a located feature.



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Component modelling – Creating features

Component created by revolving

Component created by lofting

Component created by sweeping

Study of refraction

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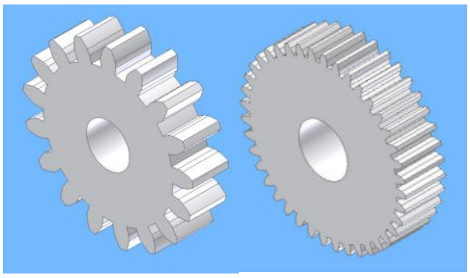
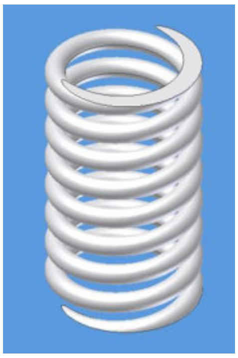
Component modelling – Some fine components





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Component modelling – Some fine components

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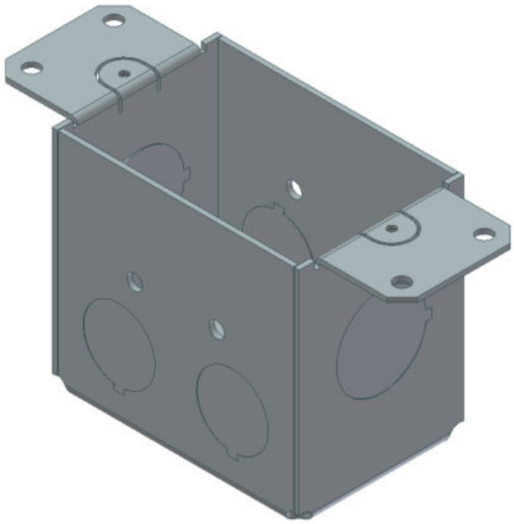
Component modelling – Plate models

An individual chapter of component modelling is represented by plate component design, plate bending, cutting, spreading, etc. by special plate commands.



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5. Attributive information and engineering calculations

Author: Olivér Papp

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Introduction

In essence, attributive information represents the properties of a Windows-based file, helping to identify, classify and group the file concerned.

It includes, for instance, an informative name, the name of the author, the subject, and key words or other important information within a file.

All 3D design softwares add extra information to components, assemblies and drawings. Some of the additional information is automatically added to the files, and they can also be extended by individual attributes.

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Grouping attributive information

Automatic. Automatic properties are generated by the application when saving files. They include the properties referring to the date of creation, the date and time of the last modification and the size of the file.

Pre-defined. Pre-defined properties already exist as early as producing the components, but the user must specify the exact values. In general, these properties are listed in a modifiable external text file and defined in the component / assembly template. Pre-defined property types include text, date, yes / no and numerical value.

Customized. Customized properties are created by the user; they include values valid for the entire document.

Configuration-dependent. Similarly to the categories above, values of this type of properties can be of several kinds, but their scope only extends to single configurations. There a number of customized attributes that are updated automatically when the component / attribute is changed, such as properties containing the weight or material of components.

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Using file attributes

Components, assemblies and drawings. File properties can be used for creating parametric legends: legends of drawings enclosed to the properties of a file and are automatically updated when the values are changed.

Assemblies. File properties can be used effectively to select and hide / show components. For example, to hide / show all the connectors or commercial goods from the assembly concerned at the same time, to select all the components created by a given designer or to group components by material.

Working drawings. File properties can be used to fill in automatically the text field of the drawing sheet, the component list, the revision block, legends and production notes.

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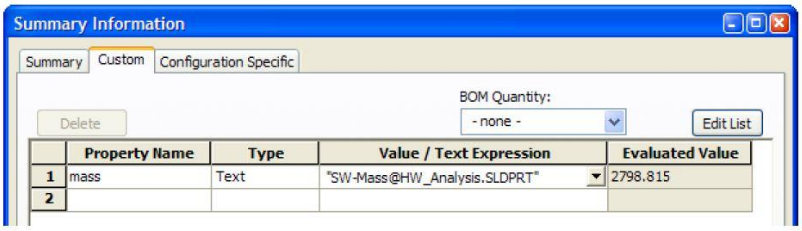
Entering customized information

File properties can be created directly within the file or by methods specified in the 3D design system.

Direct approach. File properties are created directly within the file, using Windows explorer or other file manager software.

Custom tab. By clicking on the **File** menu point in a 3D design system, you can modify and extend file properties. The customized attributes of a component / assembly can be specified through this window. Options include direct manual entry or the use of pre-defined lists.



Example for specifying mass properties as a customized file property.

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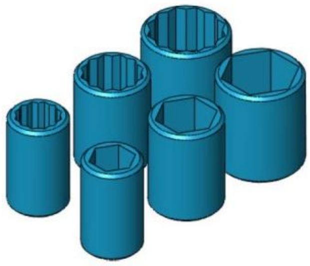
Entering customized information

Using designer's spreadsheets / tables. When producing a component set, you can also use a so-called designer's table by which you can assign customized attributes to a component (and to its configurations), for instance.



PDM / PLM system use. PDM is a Product Data Management system enabling a 3D design system to store and group data on a central server. This can be really used for team work. When components / assemblies are copied to the server, files are automatically extended by attributes, such as serial number, status, name, project name, revision. A PDM can also be configured to extend files by customized variables defined by the system administrator.

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Extracting component-related information

The two commonest examples for extracting attributive information on a 3D component are the text field of the working drawing about the component and the table including the part list of the component. Both the text field and the part list table extract and display component data by the **Assign** to property command (or an equivalent).

By a properly prepared working drawing template, a designer can save as much as an hour's work each day. As shown in the figure below, the information in blue is automatically filled in the drawing sheet when the model is inserted.

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Intelligent feature catalogue

The intelligent feature catalogue or library operation is a frequently used operation or combination of operations which is generated once and saved into the library for further use.

Library operations generally consist of operations added to the basic operation, rather than the basic operation itself. As a single model cannot include two basic operations, no basic operation can be inserted in a model already including a basic operation. However, it is possible to create a library operation containing the basic operation and to insert it in an empty model.

Users can create generally used operations such as holes or grooves and save them as library operations. A number of library operations can be used as building elements for a single model to save time and enable model consistency.

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General features

Library operations can be used for performing any of the following:

- Select configuration while inserting library operation in a model.
- Record reference to parent model.
- Edit by swapping configurations, selecting another position, etc.
- Add descriptions to references when saving library operation.
- Add drawing signs to library operation and insert them in the model together with the library operation.

  (When inserting a drawing sign into a library operation, either the drawing sign itself or the pointer must touch the operation to be saved by such operation.)

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General features

- Save helix as a library operation.
- Change sketch perspective when inserting the library operation by clicking on the arrow showing sketch perspective in the graphic area.
- Transfer visual properties specified in the library operation, e.g. patterns to the operations inserted.

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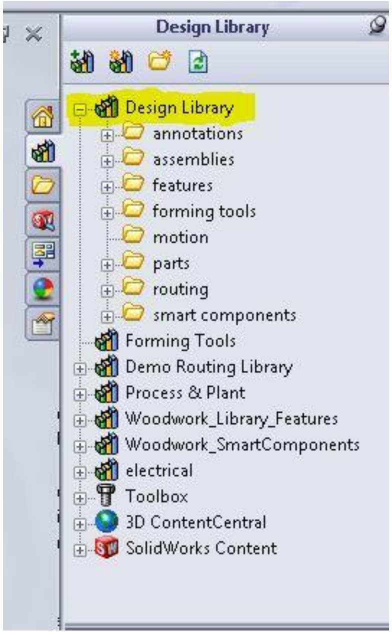
Design Library

The Design Library contains folders including reusable elements such as drawing signs, assemblies and forming tools. The Design Library manages all library operations, including the following:

- Display library operations and subfolders containing library operations.
- Preview library operation models. Design Library.
- Insert library operation into a surface element of a model or into a plane in the graphic area.



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Creating library operations

In order to create a library operation, the first task is to create a basic operation to which the designer will add the operations intended to be included in the library operation. In general, library operations are saved as separate file types. The structure of the library operation affects the way of performing the following:

- **Position** library operation on the model.
- **Edit** position of library operation.

While building up a library operation you can decide whether to include or exclude references.

References. In order to create a library operation to contain references, the library operation needs to be dimensioned in proportion to the basic model on which it was created. References generate dimensions to **position** library operations on the model.

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Creating library operations

No reference dimensions are required for library operations with surface element references e.g. fillets.

References can also be created by using relationships. For instance, a reference is created by adjusting the center point of an arc horizontally, vertically or coincidently to the sketch origin.

Location. In order to generate a library operation without references, you should create a library operation without dimensions or relationships defined in relation to the basic model. Instead of the references used for positioning the library operation, a sketch of the library operation is edited and this sketch is positioned in relation to the model.

No library operation model can be generated from a multibody model document.

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Creating library operations

Process of creating a library operation.

1. Open a new model and draw a sketch and create a basic operation from it.
2. Create the operations intended to be included in the library operation.

   After creating a model containing several extruded operations and intended to be saved as library operations, ensure that the **Link Results** option is selected. This ensures that all operations are included when the library operation is dragged onto the model.
3. Define the position of the library operation when adding it to the model:

   References. Dimensioning operations to base.

   Or

   Location. Excluding dimensions or relationships between the operation and the base.

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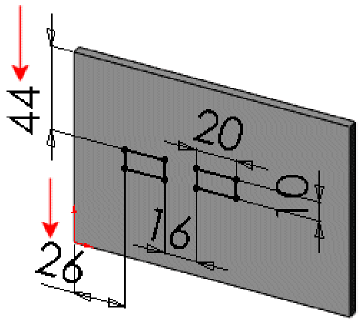
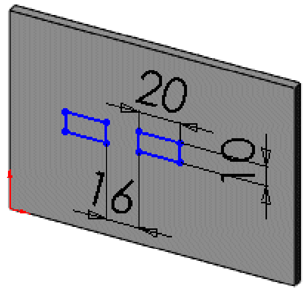
Creating library operations

Using references



Using location



4. Add further **configurations** after creating operations.
5. Close sketch and complete the following:
   - Click on the **Design Library** tab on the Task sheet.
   - Select the **Design Library** tab.
   - Select the folder where the library operation is intended to be added.

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Creating library operations

- In the Operations Manager design tree, select the operations to be saved as library operations.
- Do not include the basic operation as part of the library operation. If the basic operation is included, the base will form a part of the library operation when it is dragged onto the model.
- Hold down the **Ctrl** key and drag the operation to the bottom panel of the Task sheet.
- Subtraction of some operations may cause reconstruction errors in the library operation model due to unresolved dependencies.

6. Specify the name and one of the following (optional) in the **Save As** dialogue box:
   - Provide a description.
   - Select the **Save As Copy** option.
   - Click on the **References** key to display the **Save As With References** dialogue box.

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Engineering calculations

3D design systems can facilitate the work of construction engineers not only in modelling and spatial imaging, but also in dimensioning and engineering calculations.

The most widespread modules of engineering calculations include:

- Designing cam tool tracks.
- Inserting standard grooves.
- Supporting structure calculations.
- Dimensioning bearings.
- Inserting standard holders.
- Wall thickness analysis.
- Cogwheel / Rack design.

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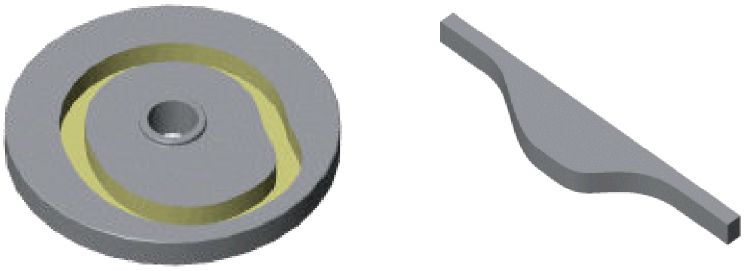
Engineering calculations (Designing cam tool tracks)

A cam track with a completely defined route of motion and a follower can be designed by it.

Options include arc or linear types and a number of motion variations. The type of the follower track can be selected. (Completely cut through or pocketed.)



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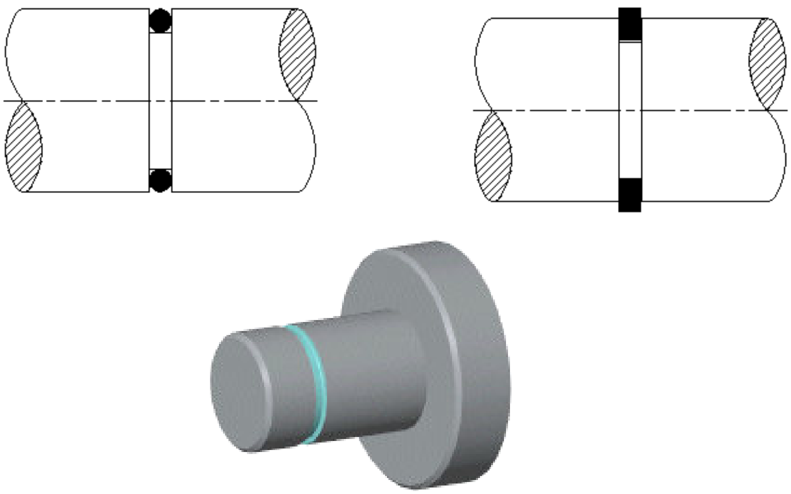
Engineering calculations (Inserting standard grooves)

Grooves dimensioned for O-rings or retainer rings of industrial standards can be adjusted to axes or cylindrical symmetric components by it.



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Engineering calculations (Supporting structure calculations)

Inclination or stress calculations can be performed by this module for the cross-section of structural figures.

Calculation process:

1. Select the load type in Rod Element Dimensions.
2. In Calculation Type, select Inclination or Stress.
3. Selected features are displayed in the data entry fields.
4. Select a rod element.
5. Select an Axis to determine the inertia torque value.
6. Enter missing features in the remaining entra field for dimensioning, and click on Solve.

   E.g. if you want to find out the rate of inclination, ensure that all fields are filled in except for inclination.

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Engineering calculations (Bearing dimensioning procedure)

The calculator can perform bearing load and simple lifecycle calculations.

Calculation process:

1. Within Bearing Dimensions, select a standard, the bearing type and a bearing available.
2. Select the measurement unit in which the result is expected.
3. Under Reliability, select an error percentage.
4. Under Capacity, select Calculated or Nominal (if capacity is known).
5. In case of equivalent loads, specify the load figure for combined radial and axial bearing loads.
6. Specify rpm for velocity.
7. Finally, click on the lifecycle calculation.

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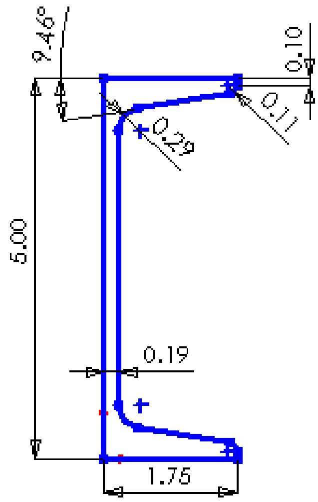
Engineering calculations (Standard structural figures)

The dimensions of standard supporting structures are fixed, designers do not need to draw them again and again. This design tool can be used to insert the dimensioned sketch of the cross-section of a structural element into any component. Dimensions are uploaded by the system from a catalogue in accordance with the applicable standard and are completely defined. After insertion, the sketch can be transformed into a body by an extrusion command.

Select the standard in the dialogue box, and the type and cross-section of the rod element based on the image in the preview window. Component properties are updated according to your selection.



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Engineering calculations (Wall thickness analysis)

The calculation tool for wall thickness analysis can be used for checking component wall thickness. It can be particularly useful in the design of die-cast plastic components or moulds of thin walls, where sudden wall thickness changes are intended to be avoided.

Wall thickness analysis can be used for the following:

– To identify thick and thin areas within a component (particularly in case of plastic components and moulds)
– To identify surfaces of a given thickness
– To detect regions which are potential hazards or sensitive to damage
– To provide design support for moulds and die-cast components

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Manufacturer's catalogues on the Internet

As 3D design systems have become increasingly widespread in industry and in everyday life as well, a demand has sprung up in construction engineers to be able to download commercially available components and assemblies from the Internet and integrate them into their designs. Initially, 3D models were provided by companies through drawing on their own resources, on CDs enclosed to their product catalogues, which is increasingly replaced by content to be downloaded from the Internet free of charge or for payment.

There are a number of businesses specialized in modelling the commercially available products of companies (to be built in target machinery) in 3D and to publish them on their webpage.

Some examples of models available this way: hundreds of SKF bearings, aluminum profiles, cooling ribs, switches, rollers, pumps and motors…

At present, there are millions of 3D models available through the Internet.

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Manufacturer’s catalogues on the Internet

3D ContentCentral

3D ContentCentral® is a free storage, configuration and download content center for 3D models, components and assemblies, as well as 2D blocks, library operations and macros.

With more than half a million registered users, this is the largest 3D online community today. After joining by simple e-mail registration the models selected can be downloaded. Anyone can upload their own models to the content center to increase the number of components available on an on-going basis.

As a supplier service, industrial component manufacturers and suppliers can also upload their components to 3D ContentCentral free of charge. The webpage provides them with further tools so that they can offer up-to-date information to construction engineers. A number of 3D design programs already provide options for direct uploads to this Internet location.

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Designer's Toolbox

The Designer's Toolbox or simply Toolbox is a library of standard connectors. This library can be used for applying pre-designed elements – standard components – in your assembly.

The Toolbox is installed together with the 3D design system, enabling users to insert commercially available fixing elements (screws, nuts, shims, etc.) into their assemblies by just a few clicks.

They can be inserted manually or by automatic recognition of hole-connector pairs.

Manual. Select the appropriate connector from the Toolbox and drag it into the assembly by holding down the left mouse button. Release the mouse button over the appropriate hole. Standard components are designed to connect automatically to the components below them by coincident and concentric constraints when being inserted in the assembly.

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Designer's Toolbox

Automatic. When a hole is created on a component, the standard dimensions of the hole are retrieved from a table by the system. The same table can be used for assigning screw types to different hole types. If the hole-connector pairs thus created are recorded in the system, the holes selected in the assembly can be filled with connectors automatically by clicking on the Quick Connector command.

Quick Connectors include a MateReferences connection setting which contains two preset constraint links. The screw can be fit into the hole by these preset constraint links. One of the constraints is **Concentric**: it is responsible for the uniaxiality of the hole and the screw. The other one is **Coincident**, which is responsible for the adjustment of the head of the screw and the surface of the hole, thus the screw is completely seated at the hole joint. You can also produce such preset connections for your components.

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Designer's Toolbox

The Designer's Toolbox includes a main component file – of standard dimensions – and a database (SWBrowser.mdb) of component dimensions and configuration data. When a standard component is inserted in a new size into the assembly, the Toolbox updates the main component file and adds a new configuration by the dimensions selected.

The Designer's Toolbox supports international standards, including ANSI, AS, BSI, CISC, DIN, GB, ISO, IS, JIS, and KS.

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CAD Book

6. Modeling of Sheet Metal Parts

Author: István Madarász

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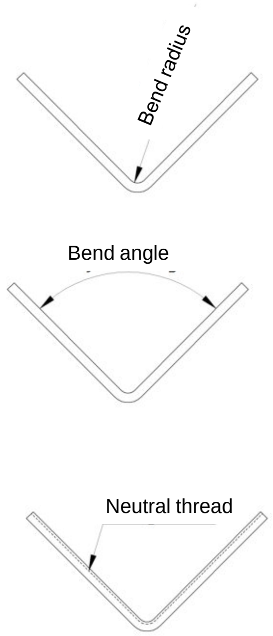
Introduction, key notes

- Material thickness
- Bend radius
- Bend angle
- Neutral line
- Neutral factor



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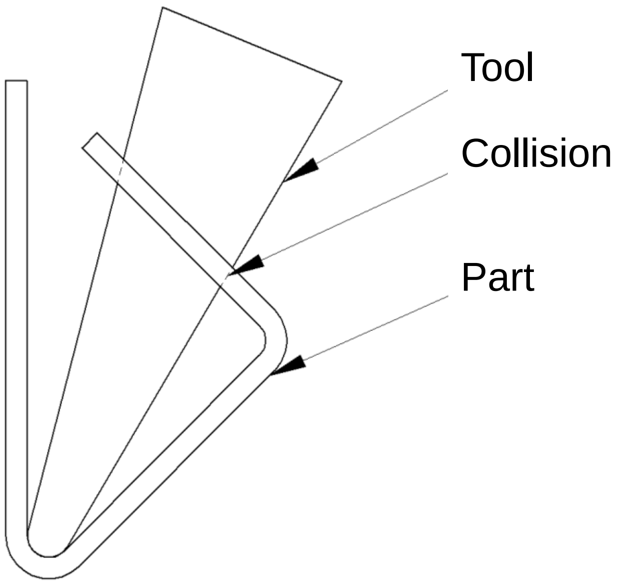
Manufacturing based design



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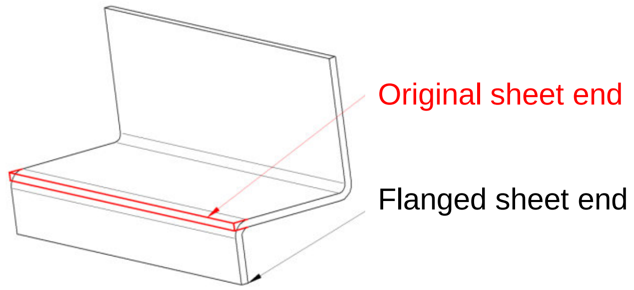
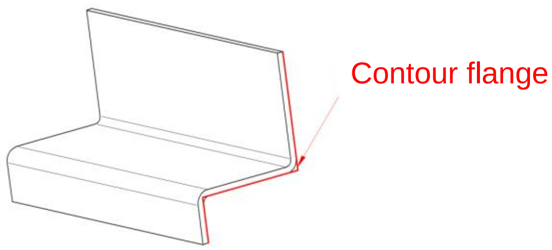
Sheet metal features I.

- Flange



- Contour flange



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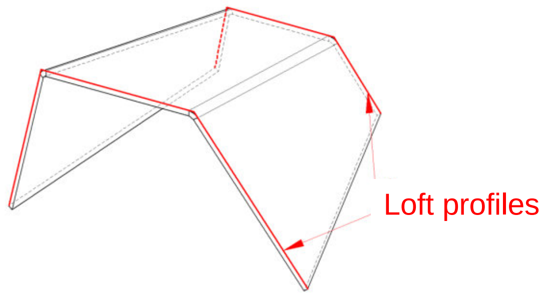
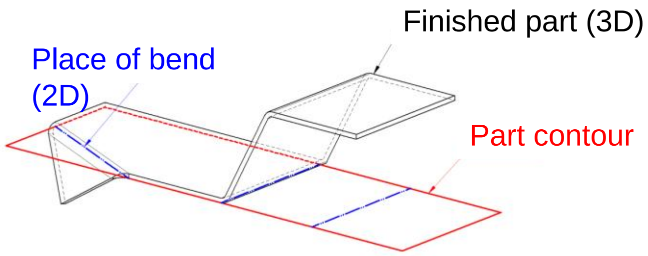
Sheet metal features II.

- Lofted flange



- Bend



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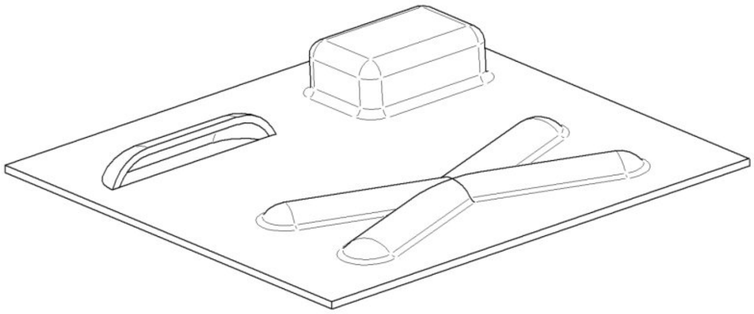
Sheet metal features III.

- Deep-draw type features



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Sheet metal features IV.

- Tab
- Cutout
  - Like part modeling
- Normal cutout
  - Cutted edges is perpendicular to sheet
- Hole
  - Like part modeling

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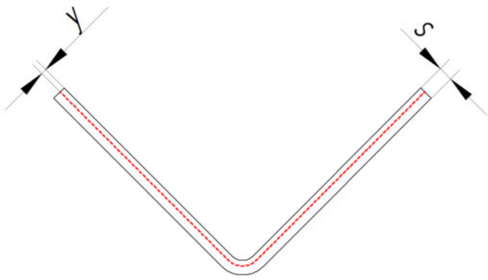
Flat pattern calculation I.

$$n = \frac{s}{y}$$



- Where:
  - n: neutral factor
  - s: material thickness
  - y: distance of neutral factor

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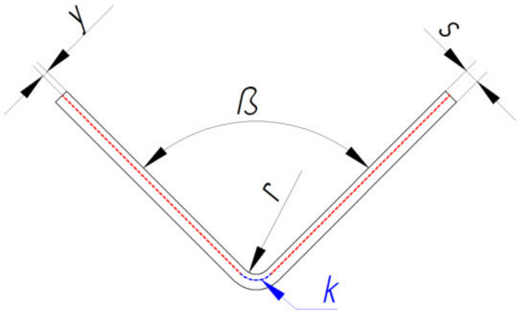
Flat pattern calculation II.

$$k = \frac{\beta \cdot \pi\left(r + (n \cdot s)\right)}{180}$$



- Where:
  - k: legth of plastic zone
  - β: bend angle
  - r: bend radius
  - n: neutral factor
  - s: material thickness

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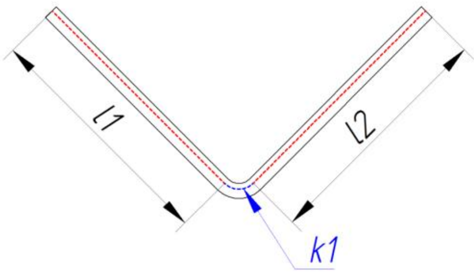
Flat pattern calculation III.

$$l = \sum_{i=1}^{n} k_i + \sum_{j=1}^{n-1} l_j$$



- Where:
  - l: length of flat pattern
  - l_j: length of a straight segments
  - k_i: legth of plastic zones

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CAD Book

7. Surface modelling

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1. Mathematical base of the surface description

- Creating interactive surface with points and curves
- Surface representation by algorithm

Continual function with two variables:

$$z = f(x, y)$$

Function of local vektor in Descartes-coordinate system:

$$\boldsymbol{P} = x \cdot \boldsymbol{i} + y \cdot \boldsymbol{j} + f(x, y) \cdot \boldsymbol{z} = \begin{bmatrix} x \\ y \\ f(x, y) \end{bmatrix}$$

**tangent sensitivity*
Solve: the implicit mode

Implicit define methode:

$F(x,y,z)=0$ $F(x,y,z)=0$ $F(x,y,z)=0$ $F(x,y,z)=0$ $F(x,y,z)=($
$F(x,y,z)=0$ $F(x,y,z)=0$ $F(x,y,z)=0$ $F(x,y,z)=0$ $F(x,y,z)=($
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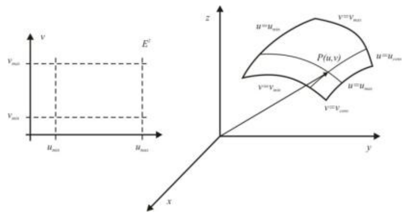
1. Mathematical base of the surface plot

To compare, let's see an example, a sphere with center in zero!

The function with variables:

$$x^2+y^2+z^2-R^2=0$$



General parametric form of surface define:

$$\boldsymbol{P}(u,v) = \begin{bmatrix} x \\ y \\ z \end{bmatrix} = \begin{bmatrix} x(u,v) \\ y(u,v) \\ z(u,v) \end{bmatrix}; \quad u_{min} \leq u \leq u_{max}\,,\; v_{min} \leq v \leq v_{max}$$

The sphere parametric representation (R= radius)

$$x(u,v) = R \cdot \cos(2\pi u) \cdot \sin(\pi v)$$
$$y(u,v) = R \cdot \sin(2\pi u) \cdot \sin(\pi v)$$
$$z(u,v) = R \cdot \cos(2\pi u) \quad u,v \in [0,1]$$

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1. Mathematical base of the surface plot

If we want to define the border curves, we have to substitute the u_{min}, u_{max}, v_{min}, v_{max} values into the parametric equation.

The border condition to adjacent patches:

$$\boldsymbol{P}(u_{max}, v) = \begin{bmatrix} x(u_{max}, v) \\ y(u_{max}, v) \\ z(u_{max}, v) \end{bmatrix} = \boldsymbol{P_2}(u_{min}, v) = \begin{bmatrix} x(u_{min}, v) \\ y(u_{min}, v) \\ z(u_{min}, v) \end{bmatrix}$$

This assure that come true the C^0 continuity criteria.

The derivated parametric function come true the C^1 continuity criteria:

$$\frac{\partial \boldsymbol{P}_{(u_{max}, v)}}{\partial u} = \frac{\partial \boldsymbol{P}_{\boldsymbol{2}(u_{min}, v)}}{\partial u}$$

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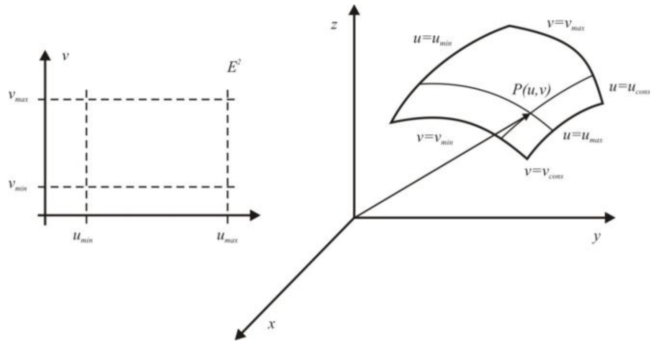
1. Mathematical base of the surface plot

This listed conditions can use on the 4 border curves and on the 4 corner points. This cornerpoints have a twisted vektor:

$$\frac{\partial^2 \boldsymbol{P}}{\partial u \partial v}$$

The patches have a normal vector. This is used for NC programs and for calculation of shadow:

$$\mathbf{N}_{(\mathbf{u},\mathbf{v})} = \frac{\partial \boldsymbol{P}_{(\,\mathrm{u},v)}}{\partial u} \times \frac{\partial \boldsymbol{P}_{(\,\mathrm{u},v)}}{\partial v}$$



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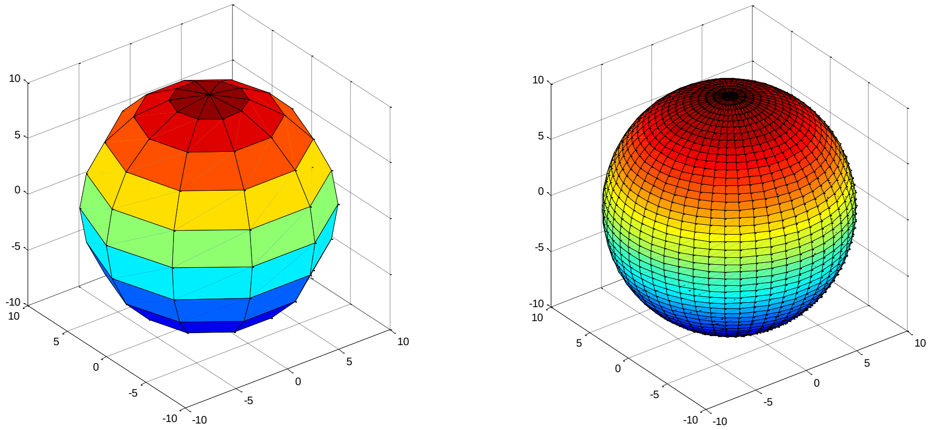
1. Mathematical base of the surface plot

- Finite number of discrate element
- m *x* n ordered parametric pair



Parametric plot of sphere in different resolution of grid

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2. Applied surfaces in the practice of the CAD

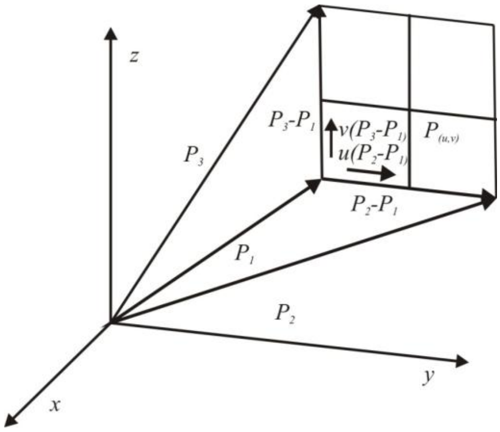
2.1. Analytical surface description

Parametric equation of a plane:

$$P(u,v) = P_1 + u \cdot (P_2 - P_1) + v \cdot (P_3 - P_1); \quad 0 \le u \le 1, 0 \le v \le 1$$

Methods to define a plane:

- Given the normal vector and a point of the plane,
- Given a curvated surface and a point where the plane is tangent.
- The plane close an optional angle with an given plane.



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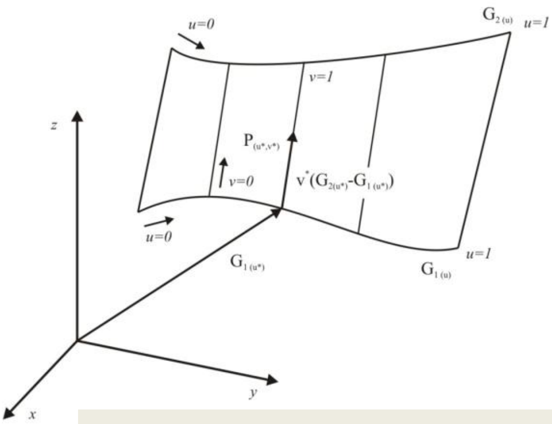
2.1. Analytical surface description



Lined surfaces:

- if a line is fixed to any point of surface - this line's all points are on this surface.

- the surface have two border curves $G_{1(u)}$ and $G_{2(u)}$

$G_{1(u)} = P_{(u,0)}$ and $G_{2(u)} = P_{(u,1)}$

The local vector between curves:

$$P_{(u,v)} = G_{1(u)} + v \cdot (G_{2(u)} - G_{1(u)})$$

Realignment:

$$P_{(u,v)} = (1-v) \cdot G_{1(u)} + v \cdot G_{2(u)}; \quad 0 \le u \le 1, 0 \le v \le 1$$

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2.1. Analytical surface description

Tabular surface:

$P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$ $P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$ $P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$ $P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$ $P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$

$P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$ $P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$ $P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$ $P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$ $P_{(u,v)} = G_{(u)} + v \cdot \mathbf{n};\ 0 \le u \le 1, 0 \le v \le 1$

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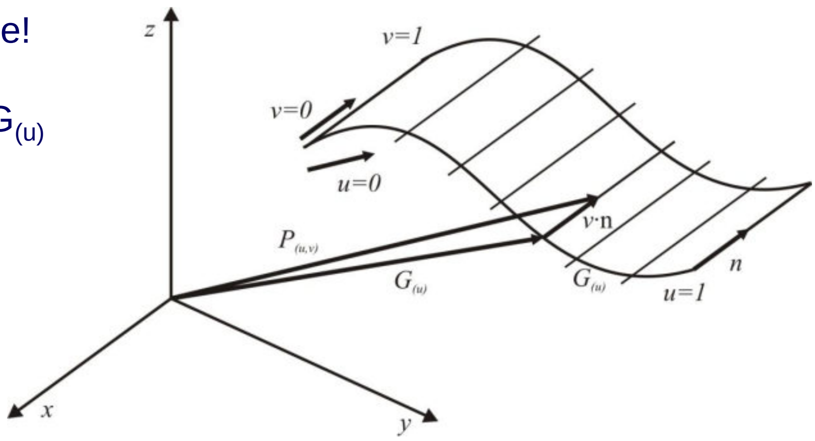
- Special lined surface!
- The control curve: $G_{(u)}$
- (n) element line by fixed direction.



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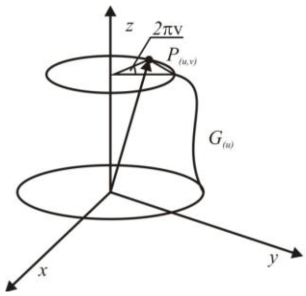
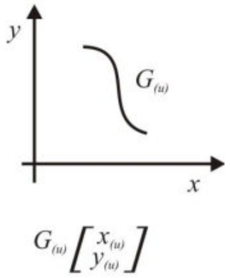
2.1. Analytical surface description



Rotated surface:

- G $_{(u)}$ profile curve,
- R $_{(u)}$ radius calculated from axis of rotation,
- The axis of rotation coinsides with the coordinate system [$r_{(u)} = x_{(u)}$].



Equation of the control curve:

$$\boldsymbol{G}(u) = \begin{bmatrix} x(u) \\ y(u) \\ 0 \end{bmatrix}; \quad 0 \le u \le 1$$

Parametric equation of the local vector:

$$\boldsymbol{P}(u,v) = \begin{bmatrix} x \\ y \\ z \end{bmatrix} = \begin{bmatrix} x(u) \cdot \cos(2\pi \cdot v) \\ y(u) \\ x(u) \cdot \sin(2\pi \cdot v) \end{bmatrix}; \quad 0 \le u \le 1\,, 0 \le v \le 1$$

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2.2. Base of the synthetic surfaces description

- Description of complicated shapes (chasses, turbina blade),
- Polynomial based approximated and interpolated parametric curves,
- Hermite – kind of surface item description.

Hermite cubic spline's arc:

$$\boldsymbol{P}(u) = \sum_{i=0}^{3} \mathbf{C}_i u^i \qquad 0 \le u \le 1$$

In scalar form:

$$x(u) = C_{3x}u^3 + C_{2x}u^2 + C_{1x}u + C_{0x}; \quad 0 \le u \le 1,$$
$$y(u) = C_{3y}u^3 + C_{2y}u^2 + C_{1y}u + C_{0y};$$
$$z(u) = C_{3z}u^3 + C_{2z}u^2 + C_{1z}u + C_{0z};$$

where: 'C' is the algebraical coefficient

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2.2. Base of synthetic surfaces description

The algebraical coefficient in form of vektor:

$$C = [C_3\ C_2\ C_1\ C_0]^T$$

T= Transposed Matrix

The parameter vektor equation:

$$\boldsymbol{u} = \left[u^3\ u^2\ u^1\ u\right]^T$$

Convert the group of equation to matrix form:

$$P(u) = \mathbf{u}^T C; \quad 0 \le u \le 1$$

The tangent's slope (Derivated Hermite equation):

$$P(u) = C_3u^3 + C_2u^2 + C_1u + C_0$$

Derivation

$$\boldsymbol{P}'(u) = 3\mathbf{C}_3u^2 + 2\mathbf{C}_2u + \mathbf{C}_1 \quad 0 \le u \le 1$$

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2.2. Base of synthetic surfaces description

If we have to define a concrete arch, the matrix can be described with substituting the border conditions.

Given: $\mathbf{P}_0$, $\mathbf{P'}_0$, $\mathbf{P}_1$, $\mathbf{P'}_1$, u=0 és u=1.

Border condition's matrix:

$$\mathbf{B} = [\mathbf{P}_0\ \mathbf{P'}_0\ \mathbf{P}_1\ \mathbf{P'}_1]^T$$

Substituted the four border conditions:

$$\mathbf{P}_0 = \mathbf{C}_0;\quad \mathbf{P'}_0 = \mathbf{C}_1;\quad \mathbf{P}_1 = \mathbf{C}_3 + \mathbf{C}_2 + \mathbf{C}_1 + \mathbf{C}_0;\quad \mathbf{P'}_1 = 3\mathbf{C}_3 + 2\mathbf{C}_2 + \mathbf{C}_1$$

The coefficients from the four equations:

$$\mathbf{C}_0 = \mathbf{P}_0$$
$$\mathbf{C}_1 = \mathbf{P'}_0$$
$$\mathbf{C}_2 = 3(\mathbf{P}_1 - \mathbf{P}_0) - 2\mathbf{P'}_0 - \mathbf{P'}1$$
$$\mathbf{C}_3 = 2(\mathbf{P}_0 - \mathbf{P}_1) + \mathbf{P'}_0 + \mathbf{P'}_1$$

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2.2. Base of synthetic surfaces description

Replaced the coefficients in to the basic equation:

$$\boldsymbol{P}(u) = \left(2u^3 - 3u^2 + 1\right)\mathbf{P_0} + \left(-2u^3 - 3u^2\right)\mathbf{P_1} + \left(u^3 - 2u^2 + u\right)\mathbf{P'_0} + \left(u^3 - u^2\right)\mathbf{P'_1}$$

The equation in matrix form (for the informatical manageability):

$$\boldsymbol{P}(u) = \left[u^3\ u^2 u\ 1\right] \cdot \begin{bmatrix} 2 & -2 & 1 & 1 \\ -3 & 3 & -2 & -1 \\ 0 & 0 & 1 & 0 \\ 1 & 0 & 0 & 0 \end{bmatrix} \cdot \begin{bmatrix} \mathbf{P_0} \\ \mathbf{P_1} \\ \mathbf{P'_0} \\ \mathbf{P'_1} \end{bmatrix} \longrightarrow \boldsymbol{P}(u) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M_{Her}} \cdot \mathbf{B}$$

- M_{Her} the Hermite matrix.

The curve's derivate in parametrical and matrixed form:

$$\boldsymbol{P}'(u) = \left(6u^2 - 6u\right)\mathbf{P_0} + \left(-6u^2 + 6u\right)\mathbf{P_1} + \left(3u^2 - 4u + 1\right)\mathbf{P'_0} + \left(3u^2 - 2u\right)\mathbf{P'_1}$$

$$\boldsymbol{P}'(u) = \left[u^3\ u^2 u\ 1\right] \cdot \begin{bmatrix} 0 & 0 & 0 & 0 \\ 6 & -6 & 3 & 3 \\ -6 & 6 & -4 & -2 \\ 0 & 0 & 1 & 0 \end{bmatrix} \cdot \begin{bmatrix} \mathbf{P_0} \\ \mathbf{P_1} \\ \mathbf{P'_0} \\ \mathbf{P'_1} \end{bmatrix} \longrightarrow P'(u) = \mathbf{u}^{\mathrm{T}} \cdot \mathrm{H_{du}} \cdot \mathrm{B}$$

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2.2. Base of synthetic surfaces description

A Hermite patches can be described by the given function of Hermite curves:

$$\boldsymbol{P}(u,v) = \sum_{i=0}^{3}\sum_{j=0}^{3} \mathbf{C}_{i,j}\, u^i v^j \qquad 0 \le u \le 1\,, 0 \le v \le 1$$

- A curve can be described with 4 vectors or 12 scalar datas,
- To describe patches 16 vectors and 48 scalar datas are needed for a precise description.
- The required parameters: 4 local vector of corner points, 2 tangent vector in all corner points (***u*** and ***v*** direction).

Matrix form of the patch's equation:

$$\boldsymbol{P}(u,v) = \mathbf{u}^T \mathbf{C} \mathbf{v}; \quad 0 \le u \le 1\,, 0 \le v \le 1,$$

where:

$$\mathbf{u} = [u^3\ u^2\ u^1\ u]^T, \text{ és } \mathbf{v} = [v^3\ v^2\ v^1\ v]^T$$

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2.2. Base of synthetic surfaces description

The algebraic coefficient:

$$C =\cdot \begin{bmatrix} \mathbf{C}_{33} & \mathbf{C}_{32} & \mathbf{C}_{31} & \mathbf{C}_{30} \\ \mathbf{C}_{23} & \mathbf{C}_{22} & \mathbf{C}_{21} & \mathbf{C}_{20} \\ \mathbf{C}_{13} & \mathbf{C}_{12} & \mathbf{C}_{11} & \mathbf{C}_{10} \\ \mathbf{C}_{03} & \mathbf{C}_{02} & \mathbf{C}_{01} & \mathbf{C}_{00} \end{bmatrix}$$

Used the already known Hermite matrix:

$$P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1$$

$$P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1$$

$$P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1 \qquad P(u,v) = \mathbf{u}^{\mathrm{T}} \cdot \mathbf{M}_{\mathbf{Her}} \cdot \mathbf{B} \cdot \mathbf{M}_{\mathbf{Her}}^{\mathbf{T}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1$$

The matrix of the border condition (16 elements):

$$B =\cdot \begin{bmatrix} \mathbf{P}_{00} & \mathbf{P}_{01} & \mathbf{P}_{v00} & \mathbf{P}_{v01} \\ \mathbf{P}_{10} & \mathbf{P}_{11} & \mathbf{P}_{v10} & \mathbf{P}_{v11} \\ \mathbf{P}_{u00} & \mathbf{P}_{u01} & \mathbf{P}_{uv00} & \mathbf{P}_{uv01} \\ \mathbf{P}_{u10} & \mathbf{P}_{u11} & \mathbf{P}_{uv10} & \mathbf{P}_{uv11} \end{bmatrix}$$

- The 4 sub-matrix include all of the local-, tangent- and twist vectors.

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2.2. Base of synthetic surfaces description

Derivates are describing the surface element’s curves:

$$P'_u(u,v) = \mathbf{u}^T \cdot \mathbf{H_{du}} \cdot \mathbf{B} \cdot \mathbf{H}^T \cdot \mathbf{v}$$
$$P'_v(u,v) = \mathbf{u}^T \cdot \mathbf{H} \cdot \mathbf{B} \cdot \mathbf{H}^T_{dv} \cdot \mathbf{v}$$
$$P''_{uv}(u,v) = \mathbf{u}^T \cdot \mathbf{H_{du}} \cdot \mathbf{B} \cdot \mathbf{H}^T_{dv} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1$$

Used with Bernstein polynomial:

$$\boldsymbol{P}(u,v) = \sum_{i=0}^{n} \sum_{j=0}^{m} \mathbf{P_{i,j}} \; \mathbf{BEZ_{i,n(u)} BEZ_{j,m(v)}}; \quad 0 \le u \le 1 \; , 0 \le v \le 1$$

P_{ij} are the control points, these creates polyeder

$P_{(u,v)}$ surface slick the polyeder of P_{ij}

Description with M_{BEZ} matrix as seen in Hermite surface defined:

$$P(u,v) = \mathbf{u}^T \cdot \mathbf{M_{Bez}} \cdot \mathbf{B} \cdot \mathbf{M}^T_{\mathbf{Bez}} \cdot \mathbf{v}; \quad 0 \le u \le 1, 0 \le v \le 1$$

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2.2. Base of synthetic surfaces description

The cubical spline based Bezier patch have 4x4 control points. These local vectors are the base of the „**B**" border condition matrix.

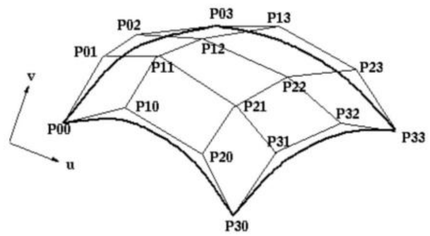
M_{BEZ} matrix:

$$\boldsymbol{M_{Bez}} = \begin{bmatrix} -1 & 3 & -3 & 1 \\ 3 & -6 & 3 & 0 \\ -3 & 3 & 0 & 0 \\ 1 & 0 & 0 & 0 \end{bmatrix}$$

Same multiply-surfaces can be describe with B-spline curves:

$$\boldsymbol{P}(u) = \sum_{i=0}^{n} \mathbf{P}_i \mathbf{N}_{\mathbf{i,k}\,(\mathbf{u})} \quad u_{min} \le u \le u_{max}\,; \quad n - k + 2 > 0$$

- maximum degree of curve is 3,
- degree of curve (k-1) depends on the control point's number,
- the parametrization depends on the spline's other preferences.



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2.2. Base of synthetic surfaces description

B-spline as a cubical equation :

$$\boldsymbol{P}(u) = \sum_{i=0}^{n} \mathbf{P}_i \mathbf{N}_{\mathbf{i},\mathbf{3}\,(\mathbf{u})} \qquad u_{min} \leq u \leq u_{max}$$

Where N $_{i3(u)}$ the cubical basis function

The basis function's general form for optional degree:

$$\boldsymbol{N}_{\boldsymbol{i},\mathbf{1}\,(\boldsymbol{u})} = \begin{cases} 1, & u_i \leq u \leq u_{i+1} \\ 0, & \text{otherwise} \end{cases}$$

$$\boldsymbol{N}_{\boldsymbol{i},\boldsymbol{k}\,(\boldsymbol{u})} = (u - u_i)\frac{N_{i,k-1\,(u)}}{u_{i+k-1} - u_i} + (u_{i+k} - u)\frac{N_{i+1,k-1\,(u)}}{u_{i+k} - u_{i+1}}$$

u_i = vertex vector

- If the distance is equal between two adjacent vertex, the spline designation is „uniform”.

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2.2. Base of synthetic surfaces description

The vertex vector:

$$u_j = \begin{cases} 0\ , & \text{ha } \ j < k \\ j - k + 1, & \text{ha } k \le j \le n \\ n - k + 2, & \text{ha } j > n \end{cases} \qquad 0 \le j \le n+k$$

The basis function is zero if it is not on the given area:

$$\sum_{i=0}^{n} N_{i,k\,(u)} = 1$$

$\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u

$\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u

$\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u

$\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u $\neq [u_i, u_{i+k+1}]$ $N_{i,k\,(u)} = 0,$ ha u

- the curve is locally controlable,
- if we increase the degree of the curve, the spline will be smoother,
- if k=n+1 (degree of the curve equal with the control point's number) the curve will be Bezier spline.

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2.2. Base of synthetic surfaces description

The surface description by B-spline:

$$\boldsymbol{P}(u,v) = \sum_{i=0}^{n}\sum_{j=0}^{m} \mathbf{P_{i,j}}\ \mathbf{N_{i,k(u)}}\mathbf{N_{j,l(v)}};\quad 0 \le u \le u_{max}\ , 0 \le v \le v_{max}$$

NURBS surfaces „Non Uniform Rational B-spline":

- not equal the distance between two adjacent vertex
- the base function is a quotient of polynoms
- we can modify the weight of control points with w_i coefficient.

The base function:

$$\boldsymbol{R}_{i,k\,(u)} = \frac{w_i N_{i,k\,(u)}}{\sum_{i=0}^{n} w_i N_{i,k\,(u)}}$$

The parametric function of NURBS's curves:

$$\boldsymbol{P}(u) = \frac{\sum_{i=0}^{n} \mathbf{P}_i\, w_i \mathbf{N_{i,k\,(u)}}}{\sum_{i=0}^{n} w_i \mathbf{N_{i,k\,(u)}}}\quad 0 \le u \le u_{max}\,;$$

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2.2. Base of synthetic surfaces description

Description of a line by B-spline:

$$\boldsymbol{P}(u) = \frac{\mathbf{P}_0\, \mathrm{w}_1(1-\mathrm{u}) + \mathbf{P}_1 \mathrm{w}_2 \mathrm{u}}{\mathrm{w}_1(1-\mathrm{u}) + \mathrm{w}_2 \mathrm{u}} \qquad 0 \le u \le u_{max}$$

Where P_0 and P_1 are two points to define the line.

If we choose unit weight, we have the general equation:

$$\boldsymbol{P}(u) = \mathbf{P}_0\,(1-\mathrm{u}) + \mathbf{P}_1 \mathrm{u} \qquad 0 \le u \le u_{max}$$

$$\boldsymbol{P}(u) = \mathbf{P}_0\,(1-\mathrm{u}) + \mathbf{P}_1 \mathrm{u} \qquad 0 \le u \le u_{max}$$
$$\boldsymbol{P}(u) = \mathbf{P}_0\,(1-\mathrm{u}) + \mathbf{P}_1 \mathrm{u} \qquad 0 \le u \le u_{max}$$
$$\boldsymbol{P}(u) = \mathbf{P}_0\,(1-\mathrm{u}) + \mathbf{P}_1 \mathrm{u} \qquad 0 \le u \le u_{max}$$
$$\boldsymbol{P}(u) = \mathbf{P}_0\,(1-\mathrm{u}) + \mathbf{P}_1 \mathrm{u} \qquad 0 \le u \le u_{max}$$

The NURBS discription for free-surfaces:

$$\boldsymbol{P}(u,v) = \sum_{\mathrm{i}=0}^{\mathrm{n}} \sum_{\mathrm{j}=0}^{\mathrm{m}} \mathbf{P}_{\mathbf{i,j}}\ \mathbf{R}_{\mathbf{i,k(u)}} \mathbf{R}_{\mathbf{j,l(v)}}; \qquad 0 \le u \le u_{max}\ , 0 \le v \le v_{max}$$

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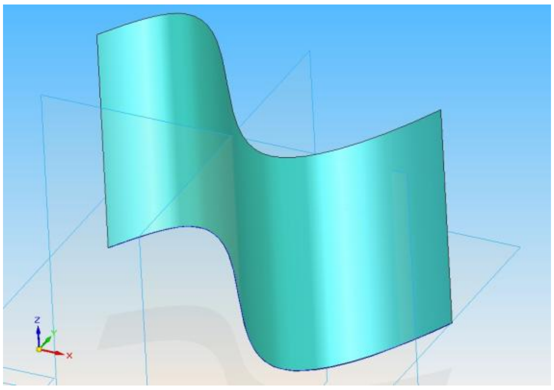
3. Typical surface-operation in the CAD systems

Input Data: points, distances, angles, relative position-constraints etc.

3.1. Extruded surfaces



Tabular surface

Control curve: from line, arc or B-spline

Operation: extrude

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3.1. Extruded surfaces

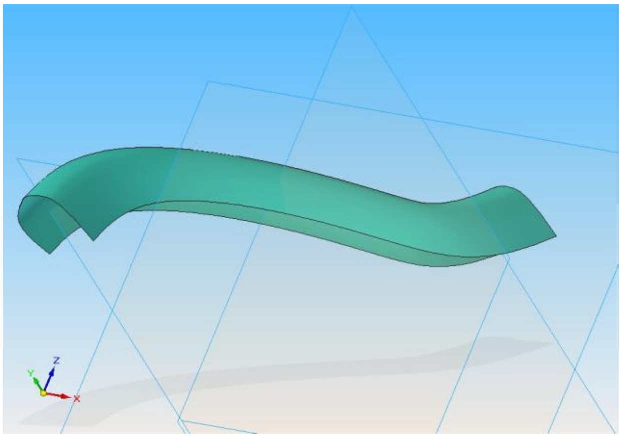
Swept surface: a special extruded surface.

Two items of this surface:

- Control curve
- Base profil

Both can be:

- Opened curve or
- Closed loop



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3.1. Extruded surfaces

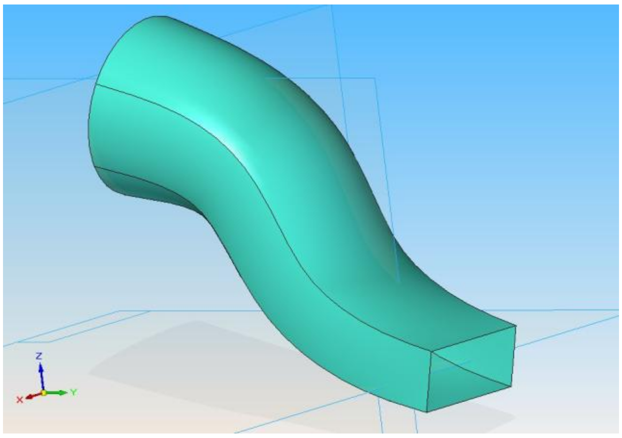
Lofted surface: Added surface between two or more profiles.

Boundary condition for transition:

- Minimum 2 profiles,
- Sufficient distance between profiles,
- Two opened or two closed profiles possible
- Specially attention to make a control point.

↓

TWIST!



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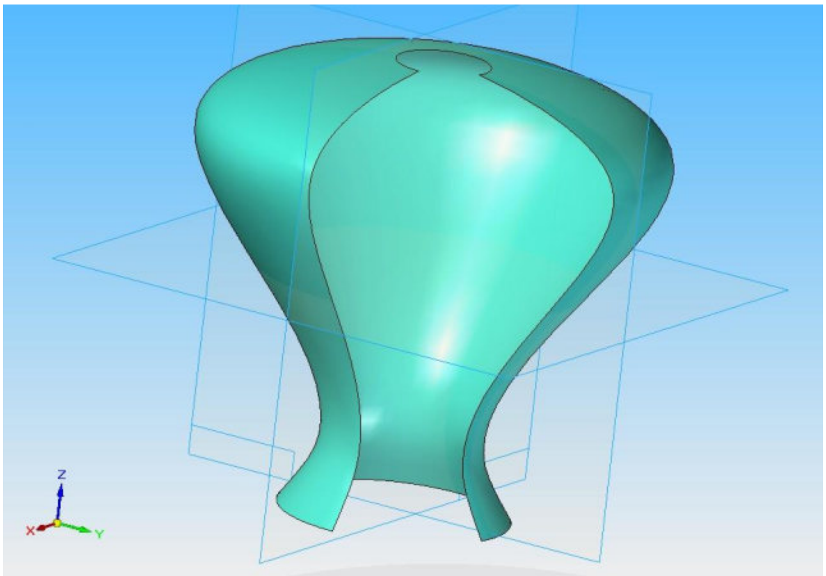
3.2. Revolved surfaces

The basis of this surface, to define a curve and a central axis.



- Opened or closed profile,
- Axis is a line (every instance),
- The curve mustn't cross the axis.

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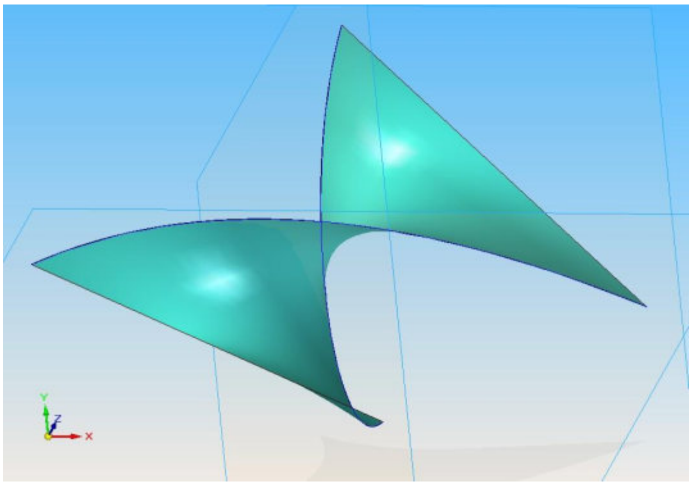
3.3. Preparation of multiple-surfaces

Mathematical background:

$$\boldsymbol{P}(u,v) = \sum_{i=0}^{n}\sum_{j=0}^{m} \mathbf{P_{i,j}}\ \mathbf{N_{i,k(u)}}\mathbf{N_{j,l(v)}};\quad 0 \le u \le u_{max}\ ,0 \le v \le v_{max}$$

- Define two B-Splines,

- Both splines must be sweep in the route, what appoint the other spline.



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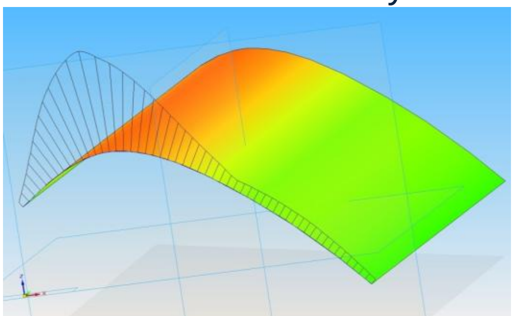
3.4. Surfaces check

- Stripes analysis – to check the continuity of surfaces
- Curvature analysis – to visualise the parameters of curve

Cylinder with zebra - stripes

Surface curvature analysis



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CAD Book

8. Engineering, assembly modelling

Author: László Rabb

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Definition of assembly

Definition

- **degree of freedom**
- removing degrees of freedom → **Assemble command**

Analysis in time (Motion analysis)

- reonom – passive or geometric relationship
- scleronom – active or kinematic relationship

Solving the non-manifold problem → more manifold occurrences (e.g. adjustable)

Types of relationships

- subordinate, commutative (face)
- superior (point, line)
- fixed relationship or including degrees of freedom → **floating**

Relationship class according to the elements

- homogeneous or heterogeneous

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History of the assembly methods

The assembly methods developed together with the modeling mode.

- ♦ CSG method, based on Boolean operations
- ♦ assembly tree, structural collecting
- ♦ parametric relationship management
- ♦ non-commutative and non-associative relationships
- ♦ direct (explicit) association, active relationships
- ♦ first steps of parametric, active relationships using and non-manifold assemblies
- ♦ synchronous technology, that integrates direct and parametric methods, based on live rules
- ♦ *future– hybrid technology that can handle non-linear properties as well (e.g. changing density with deformation)*
- ♦ *future – technology that applies complex kinematic relationships (e.g. vascular moving)*

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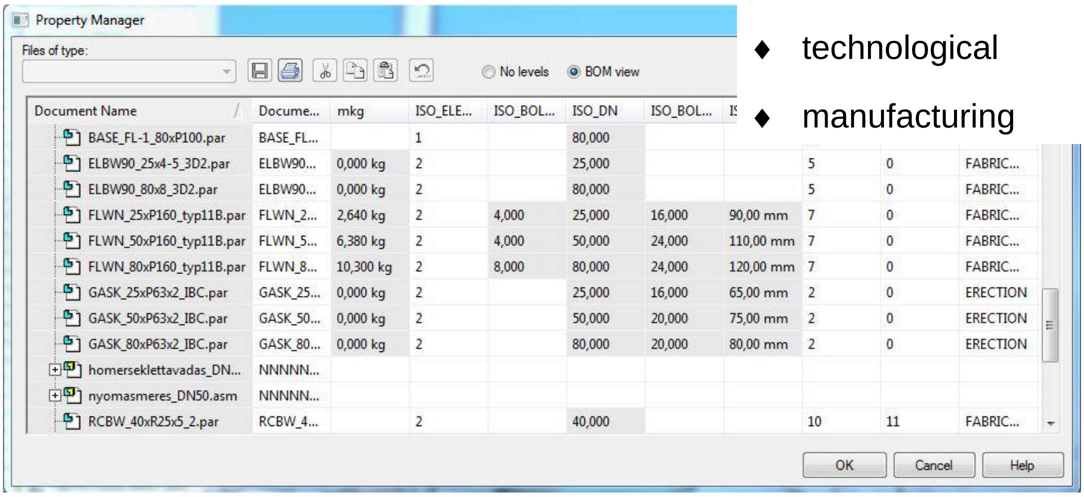
Non geometric parameters of the assembly

The assembly model consists of two data-groups: geometric data and non geometric attribute data → Consistent model

Non geometric data:

- ♦ administrative
- ♦ technological
- ♦ manufacturing



Document Name	Docume...	mkg	ISO_ELE...	ISO_BOL...	ISO_DN	ISO_BOL...	IS			
BASE_FL-1_80xP100.par	BASE_FL...		1		80,000					
ELBW90_25x4-5_3D2.par	ELBW90...	0,000 kg	2		25,000			5	0	FABRIC...
ELBW90_80x8_3D2.par	ELBW90...	0,000 kg	2		80,000			5	0	FABRIC...
FLWN_25xP160_typ11B.par	FLWN_2...	2,640 kg	2	4,000	25,000	16,000	90,00 mm	7	0	FABRIC...
FLWN_50xP160_typ11B.par	FLWN_5...	6,380 kg	2	4,000	50,000	24,000	110,00 mm	7	0	FABRIC...
FLWN_80xP160_typ11B.par	FLWN_8...	10,300 kg	2	8,000	80,000	24,000	120,00 mm	7	0	FABRIC...
GASK_25xP63x2_IBC.par	GASK_25...	0,000 kg	2		25,000	16,000	65,00 mm	2	0	ERECTION
GASK_50xP63x2_IBC.par	GASK_50...	0,000 kg	2		50,000	20,000	75,00 mm	2	0	ERECTION
GASK_80xP63x2_IBC.par	GASK_80...	0,000 kg	2		80,000	20,000	80,00 mm	2	0	ERECTION
homerseklettavadas_DN...	NNNNN...									
nyomasmeres_DN50.asm	NNNNN...									
RCBW_40xR25x5_2.par	RCBW_4...		2		40,000			10	11	FABRIC...

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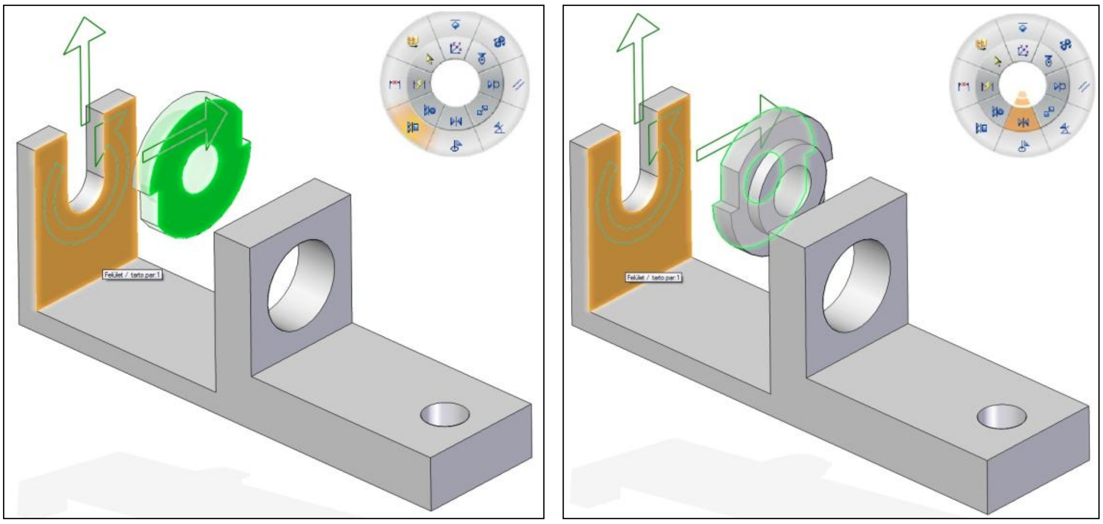
Assembly – geometric relationships

Mate and planar align / subordinate-3

Matching the face normals: same or opposite directions



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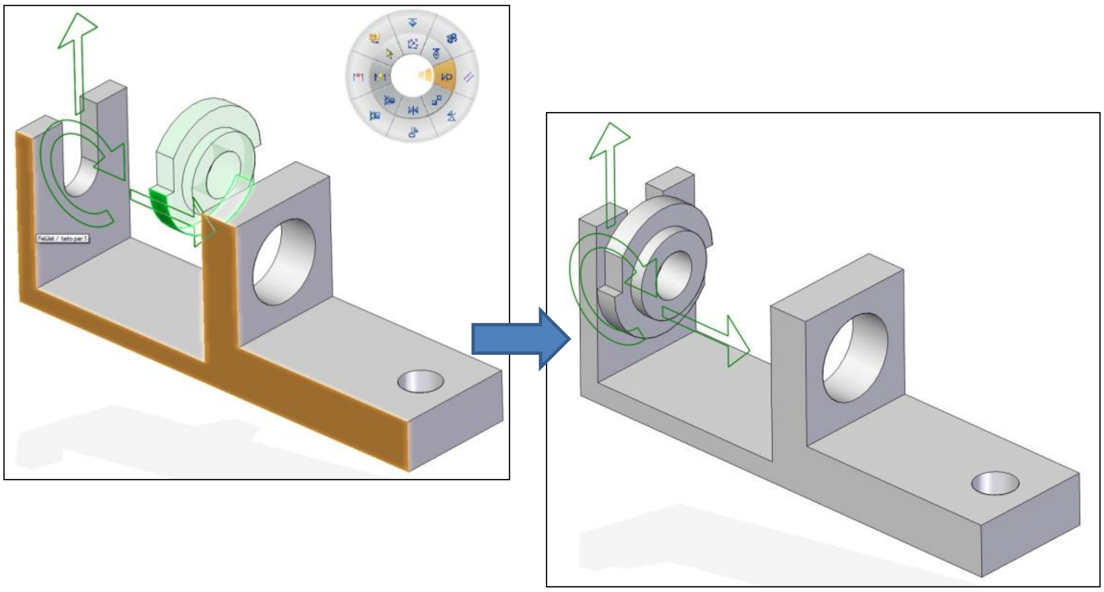
Assembly – geometric relationships

Tangent relationship/ subordinate-3



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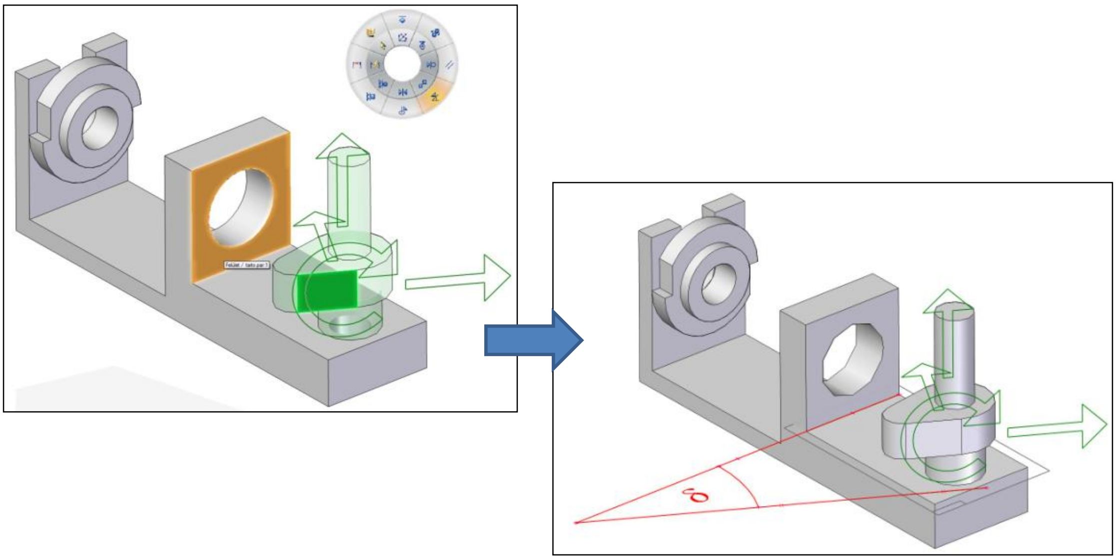
Assembly – geometric relationships

Angle relationship/ subordinate-2



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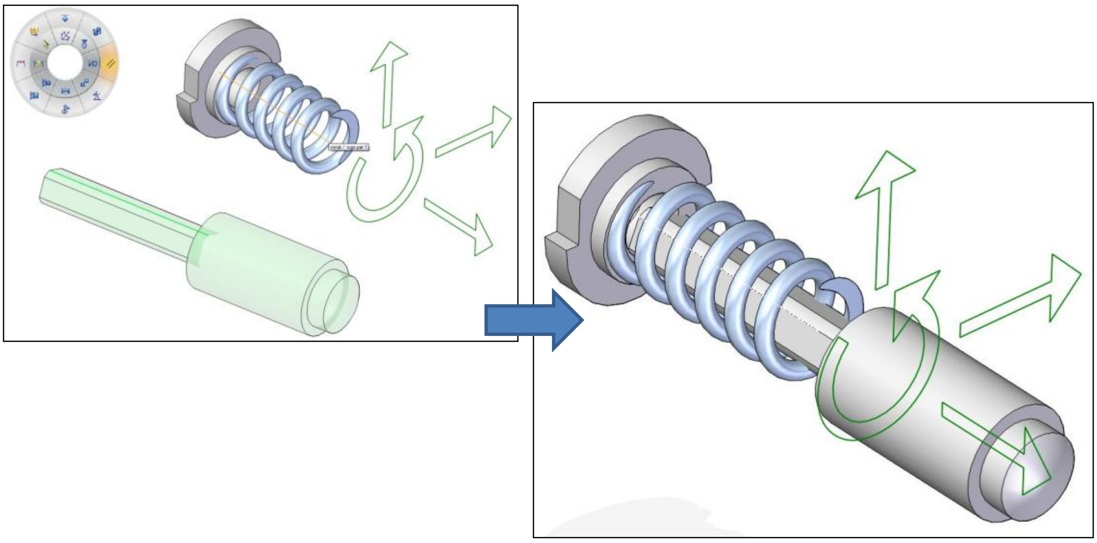
Assembly – geometric relationships

Parallel relationship/ subordinate-2

(sometimes the parallel edges can be collinear, in this case: superior-4)



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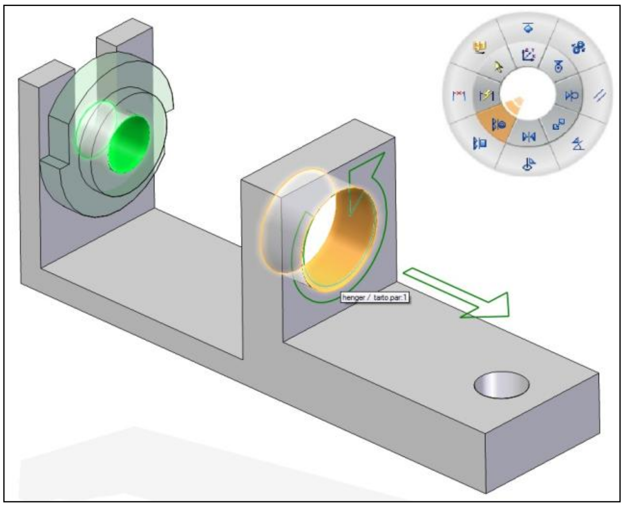
Assembly – geometric relationships

Axial relationship/ superior-2

(derives from parallel)



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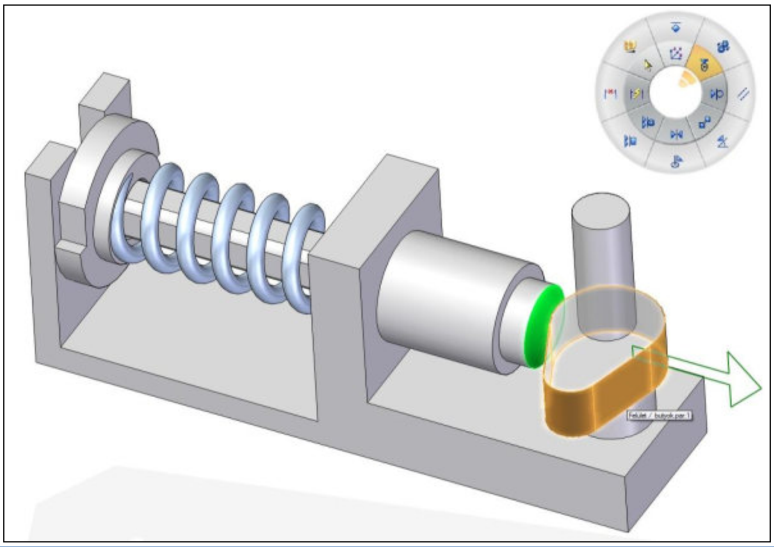
Assembly – geometric relationships

Cam relationship / superior-5

(basically active, but can be used as passive feeds back to tangent relationship)



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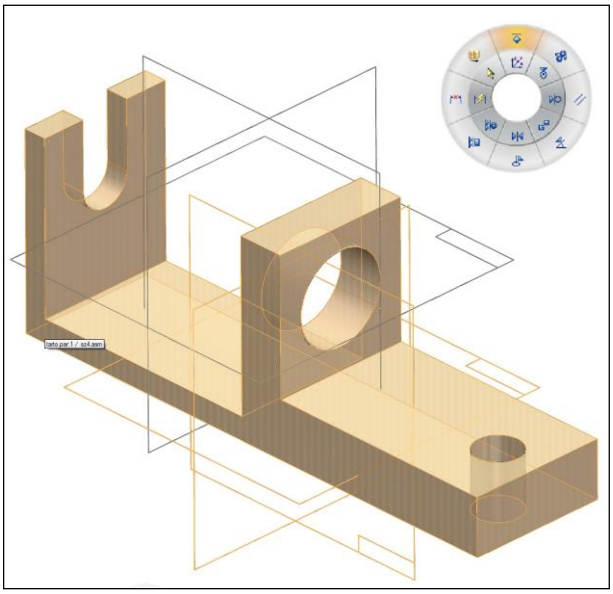
Assembly – geometric relationships

Ground relationship and coordinate system align / superior-6

(removes all degrees of freedom)



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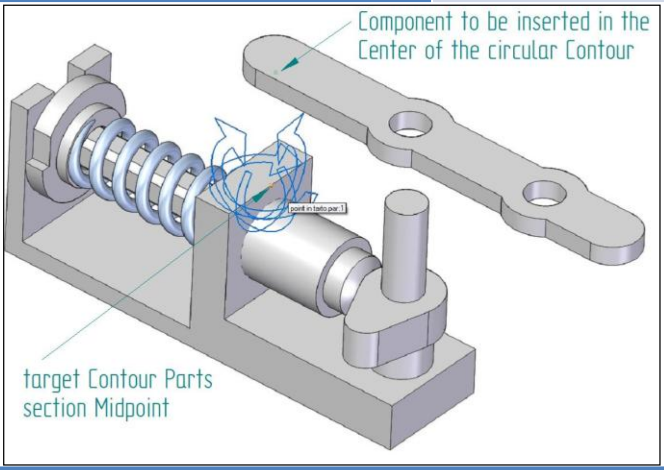
Assembly – geometric relationships

Heterogeneous relationships

Elements	Degree number	Sorting
point-point (ball socket)	3	subordinate
point-line (ball socket+slider)	2	subordinate
point-face (ball socket+pair of sliders)	1	subordinate
line-line → parallel (ball socket+slider)	4	superior
line-face → not applicable		



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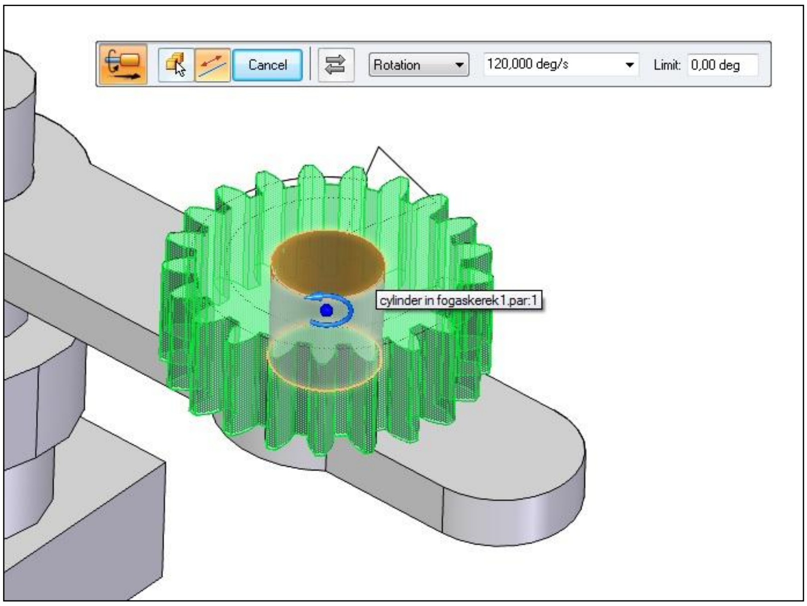
Assembly – kinematic relationships

Source relationship / superior-1

(motor)



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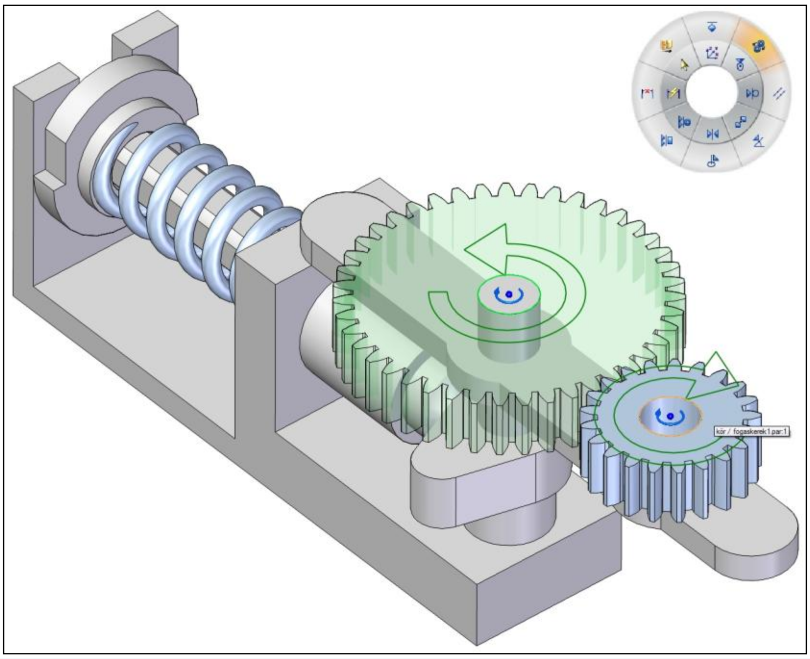
Assembly – kinematic relationships

Indirect – Gear relationship / superior-4



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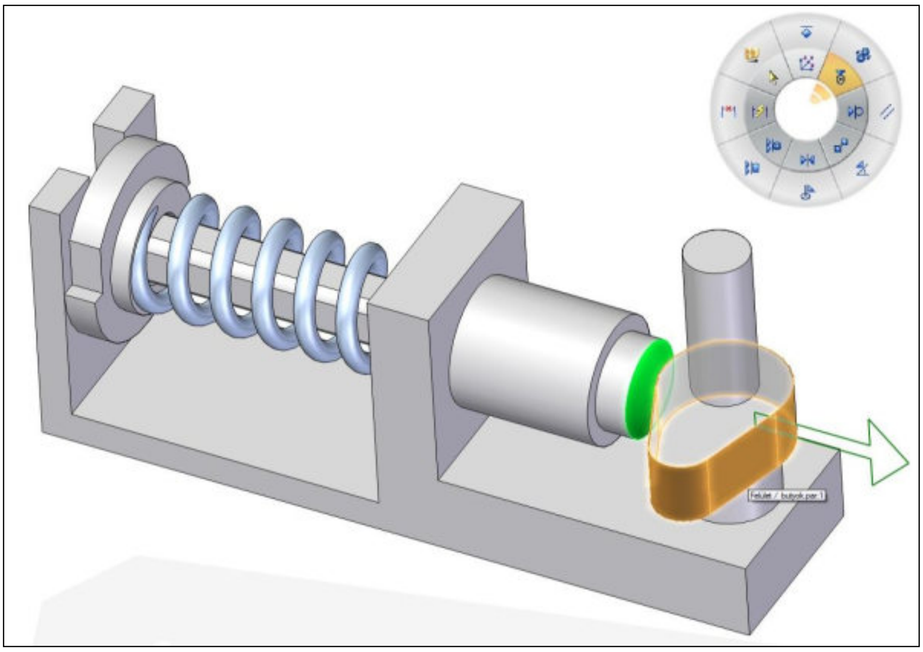
Assembly – kinematic relationships

Indirect – Cam relationship / superior-5



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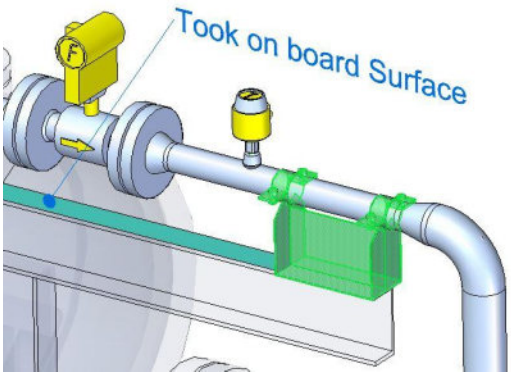
Assembly – other relationships

Non geometric relationships

- technological relationship
- structure relationship

Special indirect relationships

- Indirect subassembly relationship



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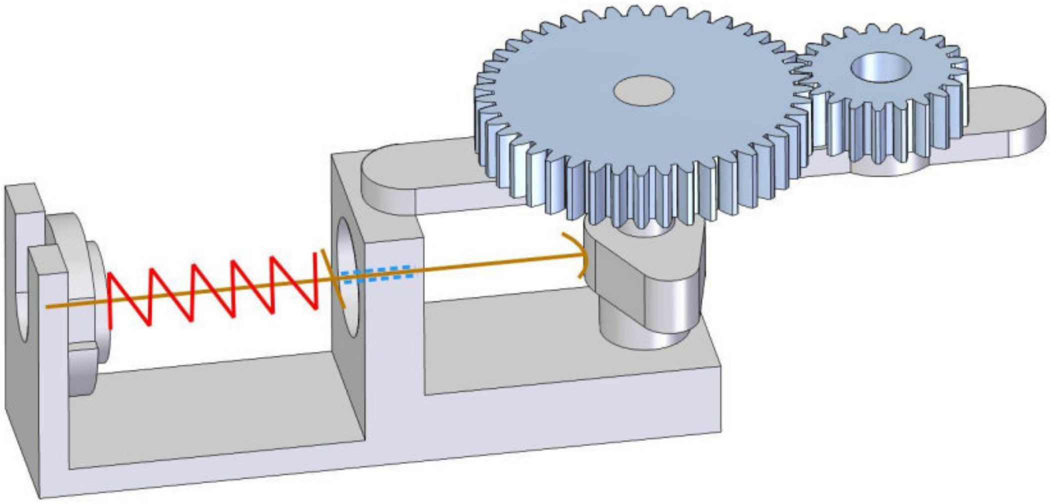
Assembly – other relationships

- systems library
- reassembled part
- virtual relationship



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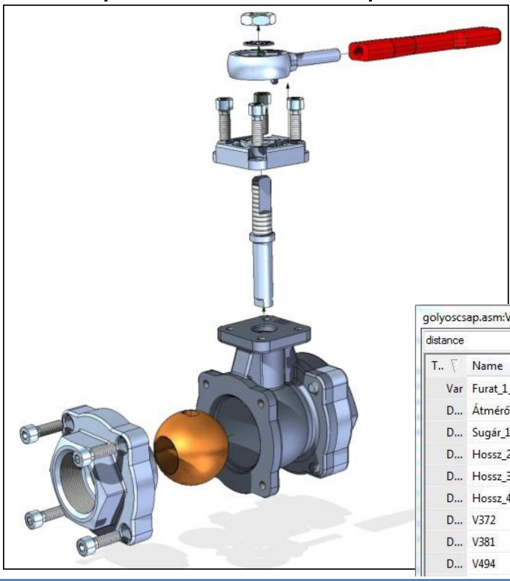
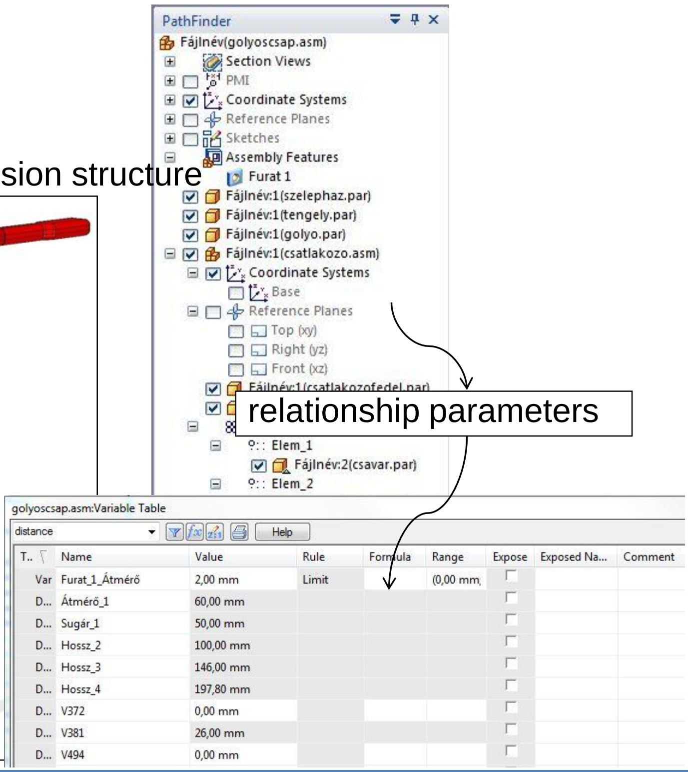
Assembly structure

Assembly structure

- functional hierarchy
- exploded view → explosion structure





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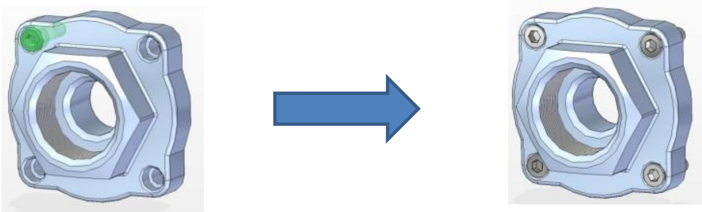
Assembly - operations

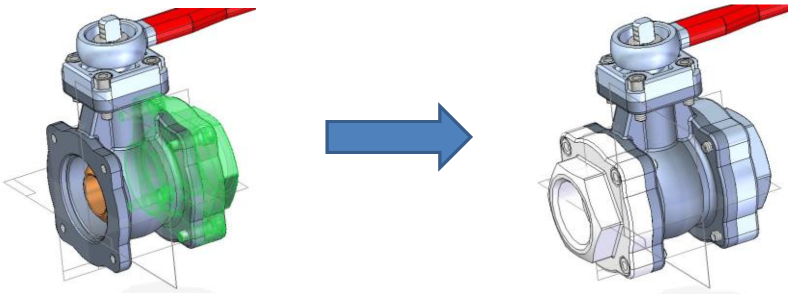
Modification, transformation

Editing

- pattern



- mirror



Additive operations (pl. explosion history)

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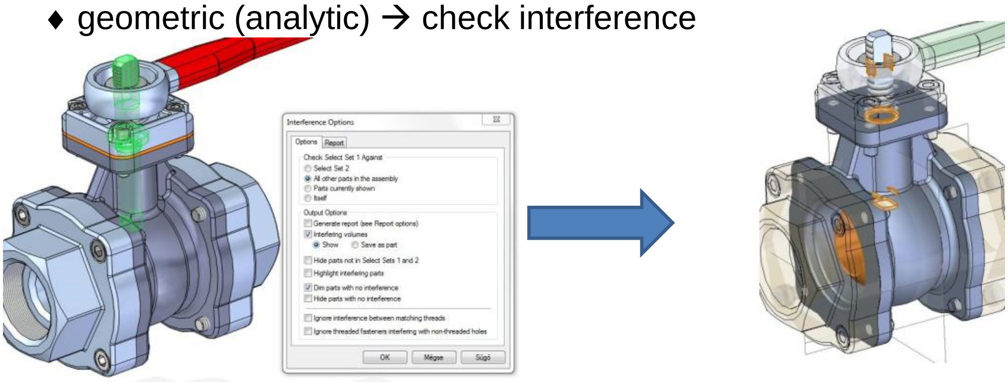
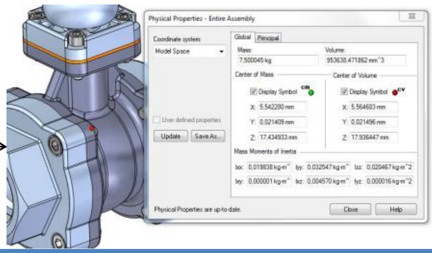
Assembly - operations

Inspection

- geometric (analytic) → check interference



- kinematic → adjustable assemblies
- physical → material and physical properties
- statistic → BOM



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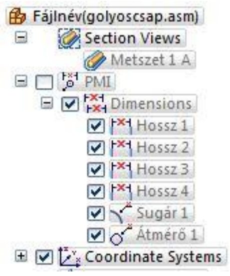
Assembly - operations

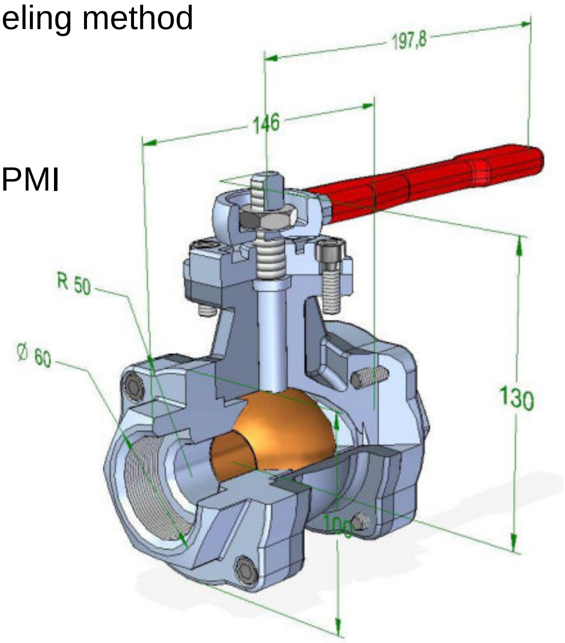
Application operations

- ♦ embedded or referenced modeling method

Additive entities

- ♦ assembly sketches
- ♦ manufacturing data, views → PMI





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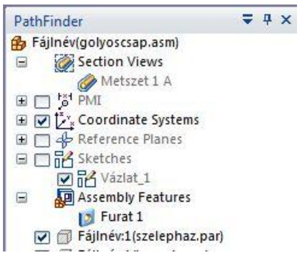
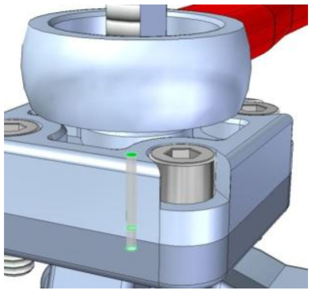
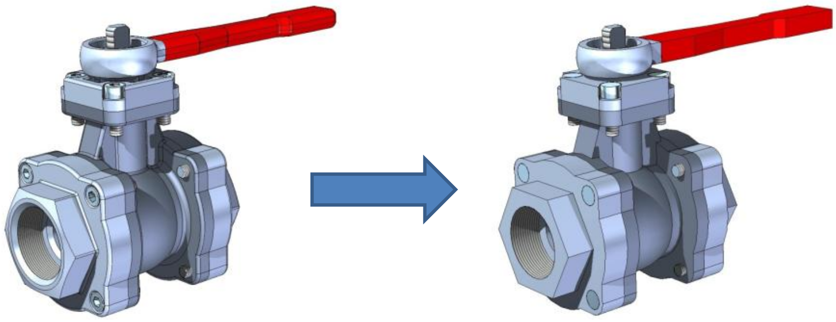
Assembly - operations

- assembly features →material removing
  → welding





- Simplified model →redundant model, fast using



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Assembly – effects of model building

The modeling technology determines the possible relationships to be used. The assembly techniques depend on the modeling methods.

- CSG method → Boolean connections in the relationship chain, fix dimensions
- Parametric method → the relationships can be defined with parameters, there is an assembly structure, in which the parameters can be re-defined
- direct method → the connection is local geometry dependent, there is no assembly structure, the assembly can only be interpreted relatively to the latest state
- synchronous method → local geometry dependent, but srtuctured with live rules, integrated consistently with the relationships, there is no assembly structure, but there are procedural connections
- hybrid method → uses the synchronous and the parametric methods simultaneously, consistently. The methods can be used and switched freely, and can contain any proportion of synchronous and parametric elements.

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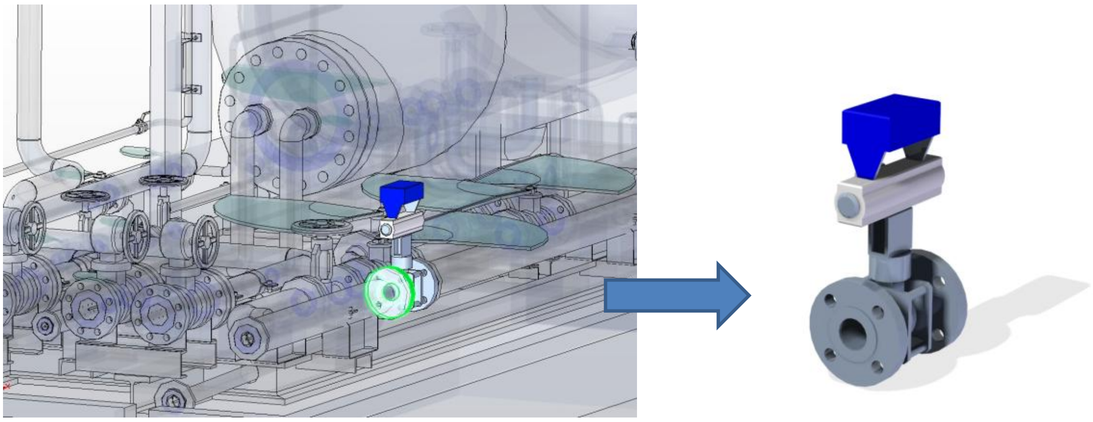
Assembly – effects to the design

The traditional design method is the bottom-up method;

part → subassembly → assembly → product

The effects of the environment can be considered better if the engineer knows the parameters of the containing environment. The engineering logic is the top-down method;

product → adequate assembly structure → part



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CAD Book

9. Kinematical Analysis in CAD Environment

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Introduction

The purpose of the computerised analysis of mechanisms course is to acquaint you with the structure of mechanisms, to interpret the various techniques of position and track calculation, to perform velocity analysis, and to define the velocity of the combined body in motion. Building on basic knowledge, the purpose is to teach model creation steps for kinematic analysis and finally to demonstrate possible application with case studies.

Keywords: mechanism, simulation, joint, kinematical analysis

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Main Topics

- Basis of Mechanisms
- 3D Modell Building for kinematical analysis
- Case study

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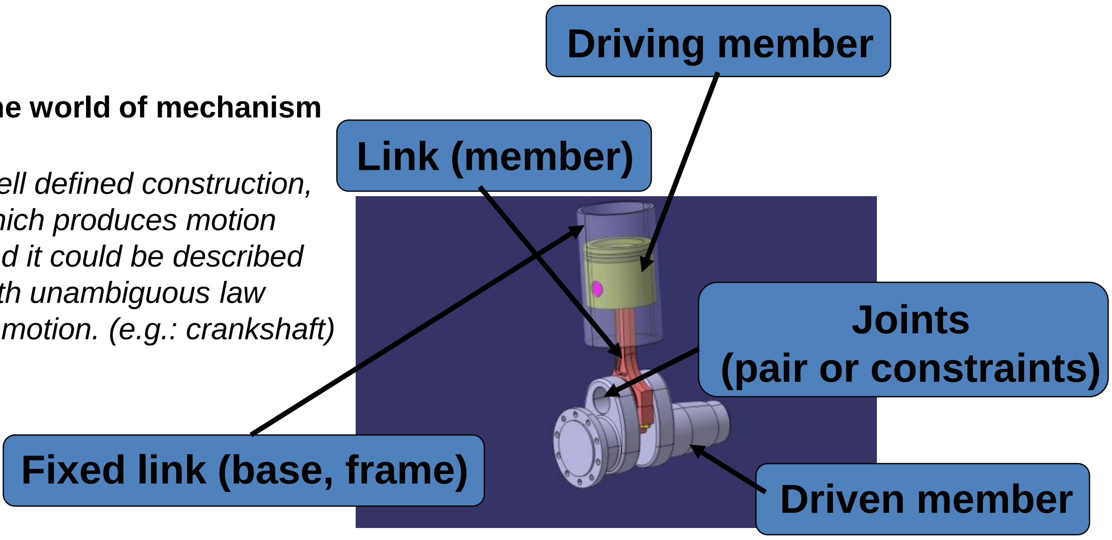
Basis of Mechanisms

The world of mechanism

Well defined construction, which produces motion and it could be described with unambiguous law of motion. (e.g.: crankshaft)



Structural construction of a crank-shaft mechanism

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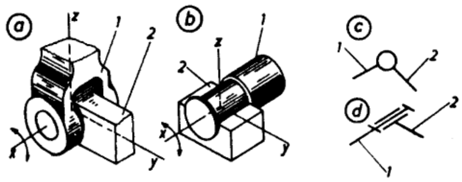
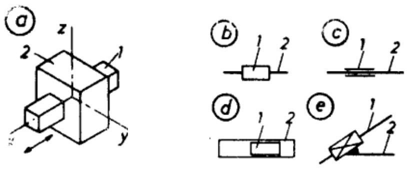
Basis of Mechanisms

Kinematical pairs (joints) and their character

DOF definition



Sketch and construction of revolute joint



Sketch and construction of translational joint

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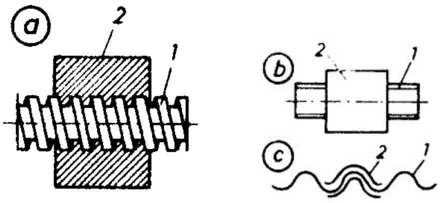
Basis of Mechanisms



Sketch and construction of screw joint

Sketch and construction of spherical joint

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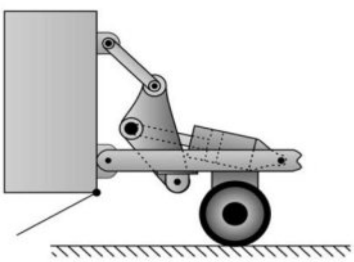
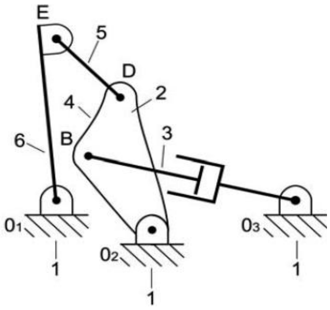
Basis of Mechanisms

Kinematical sketch

Characterize the construction and kinematical features – enable to define
-track
-velocity
-acceleration
-forces.





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Basis of Mechanisms

How to define DOF of a mechanism?

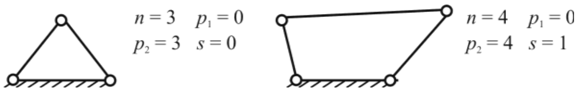
Degree of freedom ⟷ **Constraint**

$$s = 3(n-1) - 2p_1 - p_2$$

Kuztbach criteria (planar mechanism)

Number of input parameters which must be controlled independently in order to bring the device into a particular position.

n – number of links
p_1 – single-degree-of-freedom pairs
p_2 - two-degree-of-freedom pairs



some usual kinematical strucrure and their calculation

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Basis of Mechanisms

How to define DOF of a mechanism?

Grübler equation

in case of: (p1=0, s=1)

$$3n - 2p_2 - 4 = 0$$

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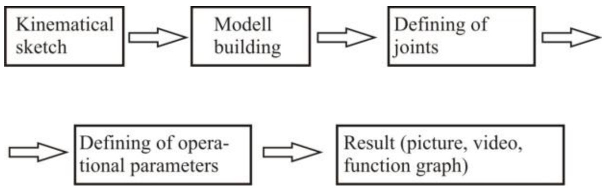
3D Modell Building for kinematical analysis

Criterias of 3D modell creation



Workflow of computer aided kinematical simulation

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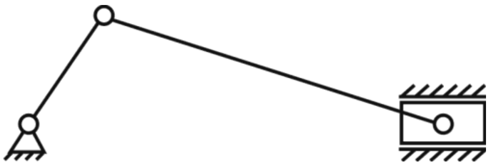
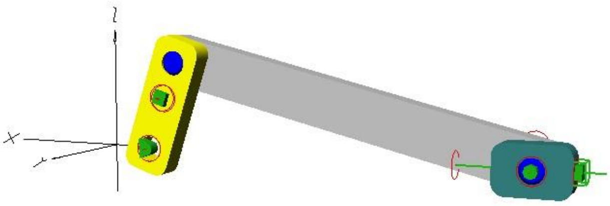
3D Modell Building for kinematical analysis

Simplifyed modells

Absolute and relative coordinate systems - how to apply them when defining the joints and members.





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3D Modell Building for kinematical analysis

Importing 3D CAD solid modells

Simulating the real structure – force, load and stress analysis

Look for number of parts (just the most important ones from the point of view of mechanism operation.

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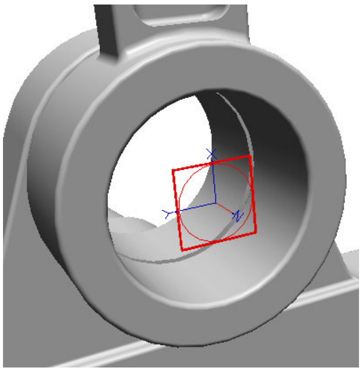
3D Modell Building for kinematical analysis

How to define joints

The basic joints type are build in the softwares.



Defining the centre of hole with an auxiliary coordinate system for revolute joint definition.

Do not forget to define the connection to the background in the necessary parts.

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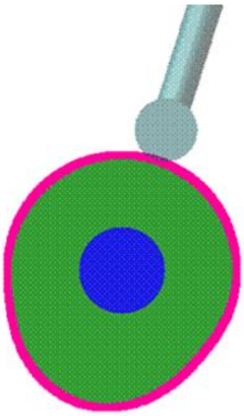
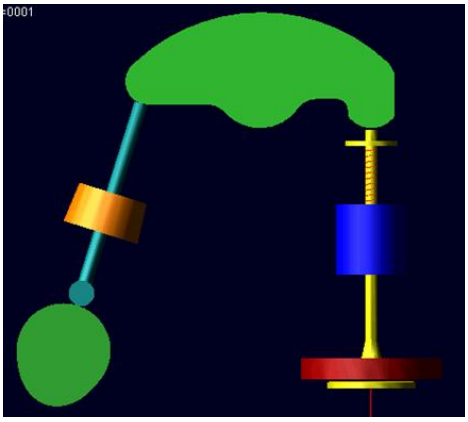
3D Modell Building for kinematical analysis

Special connections (joints).

Point-curve constraints (enable or disable lift off).
Define spline on the necessary surface.





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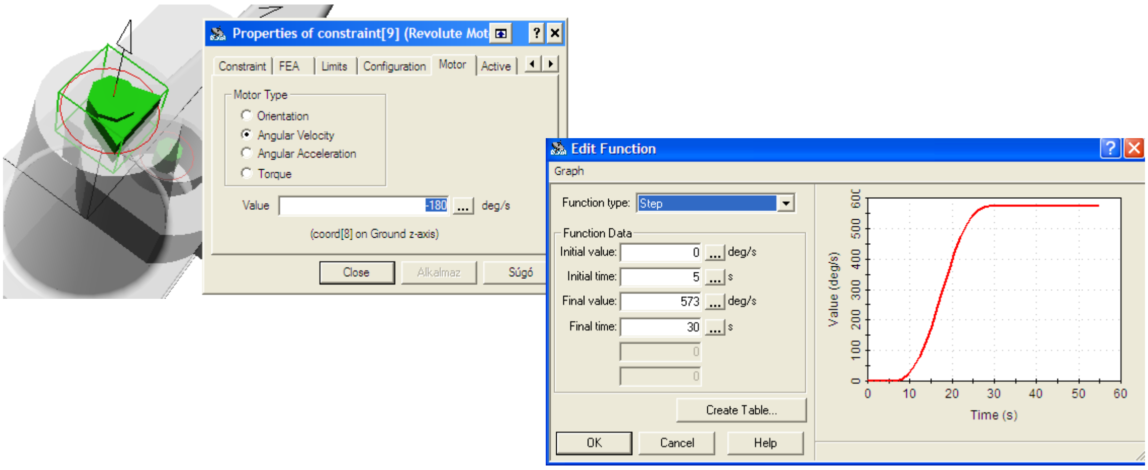
3D Modell Building for kinematical analysis

Defining the operating parameters.



A possible set of driving parameters

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3D Modell Building for kinematical analysis

Visualisation of results.

Track
Velocity or acceleration functios
Picture
Animation (video)

Criterias of a working modell:
- proper definition of joints (constraints),
- definition of the bakcground constraints,
- definition of the driving part and parameters.

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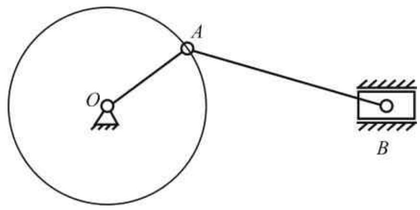
Case study

Analysis of a crank-shaft mechanism

1. Kinematical sketch



2. Modell creation (building the 3D parts)

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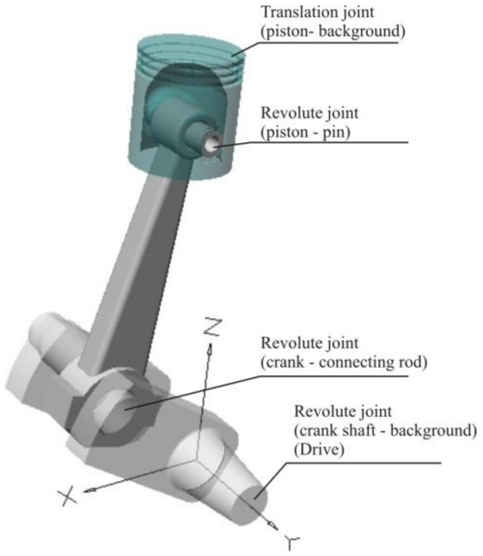
Case study

Analysis of a crank-shaft mechanism

3. Definition of the joints



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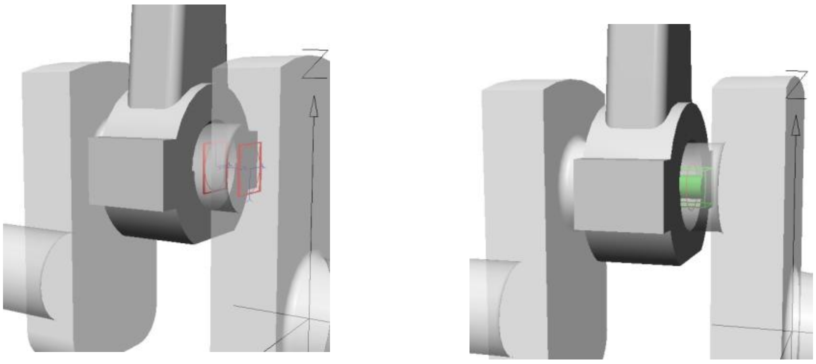
Case study

Analysis of a crank-shaft mechanism

Process of revolute joint creation



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Case study

Analysis of a crank-shaft mechanism

Definition of translation joint
- place
- connected parts
- direction

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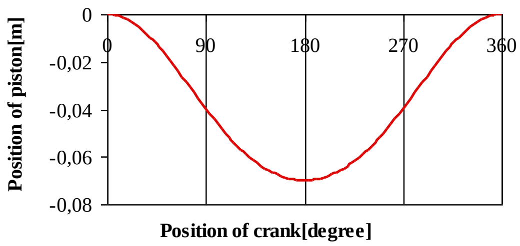
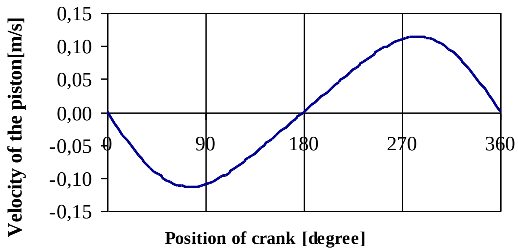
Case study

Analysis of a crank-shaft mechanism

Results





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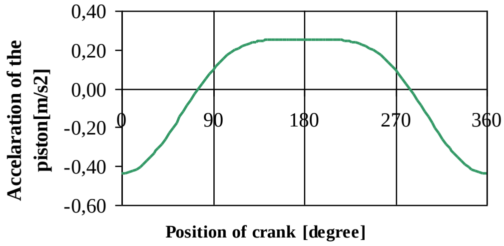
Case study

Analysis of a crank-shaft mechanism

Results



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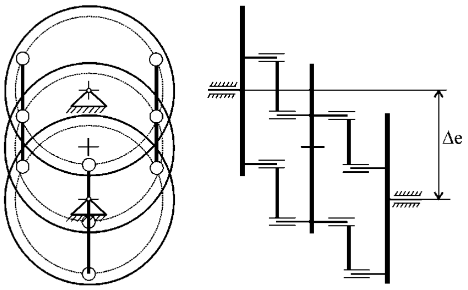
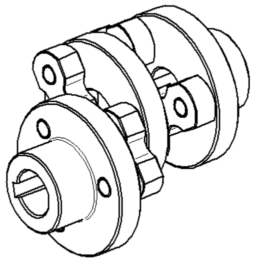
Case study

Analysis of a Smidt-coupling

1. Kinematical sketch



2. 3D modell



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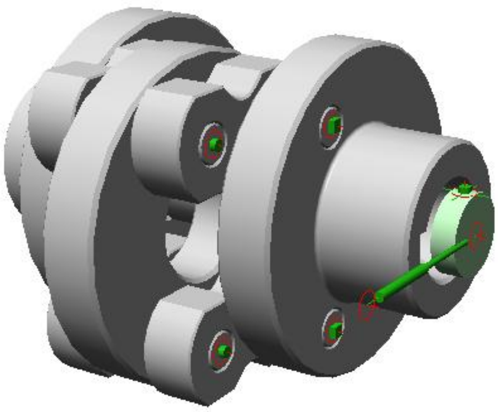
Case study

Analysis of a Smidt-coupling

3. Definition of joints
- revolute joints
- connection to the background
- radial positioning



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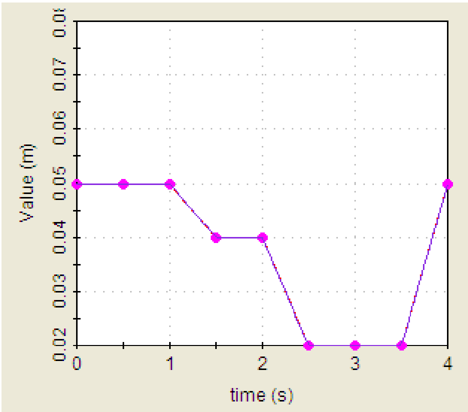
Case study

Analysis of a Smidt-coupling

Operational parameters
- RPM
- Torque
- radial displacement



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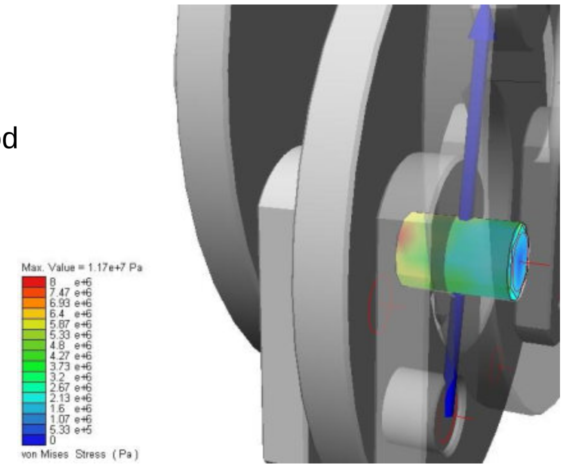
Case study

Analysis of a Smidt-coupling

Results:
- simulation (workability)
- RPM of the driven shaft
- force analysis of the connecting rod
- analysis of bolts load



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CAD Book

10. Engineering drawing in CAD environment

Author: Dr. Balázs Mikó
miko.balazs@bgk.uni-obuda.hu

K.L. Narayana, P. Kannaiah, K. Venkata Reddy Machine drawing New Age, New Delhi 3rd ed. 2006. ISBN 978-81-224-2518-5

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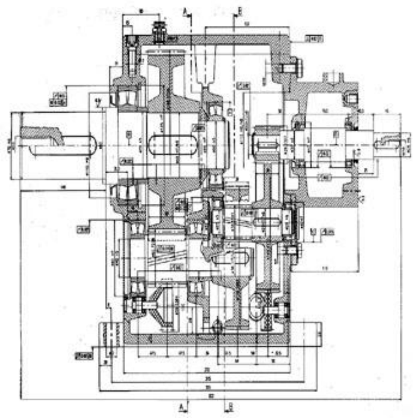
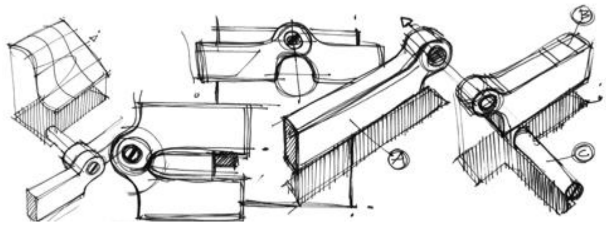
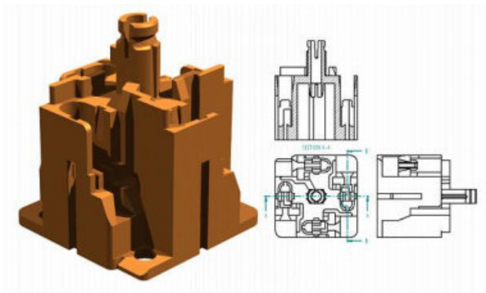
Part description

Speach
Text
Free sketch
Engineering drawing
CAD model







There are several possibilities to describe a machine parts' geometry. Beside the speech, the text and free hand sketch the typical media of it is the engineering drawing. The fundaments of the engineering drawing is constantly in the last hundred years, but the CAD systems added lot of possibilities to generate them and extend the contains of them.

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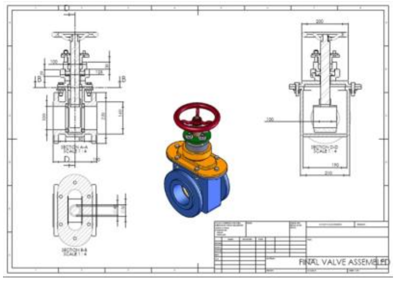
Modelling process

Part modelling



Engineering drawing

Assembly

Drawing

- Based on 3D CAD model
- Preparation in model space
  - Define view directions
  - Define section planes
  - Set parameter table

In a CAD system the engineer create a 3D virtual model of a part, and he can create an assembly with using of more parts. The engineering drawing will generated based on this model, so the media of the design is the 3D virtual space, not the drawing sheet. In order to generate the drawing, some definitions are added in the virtual space, like the view directions, sections and the contain of the parameter table.
Because of the associativity the drawing will show the actual state of the model.

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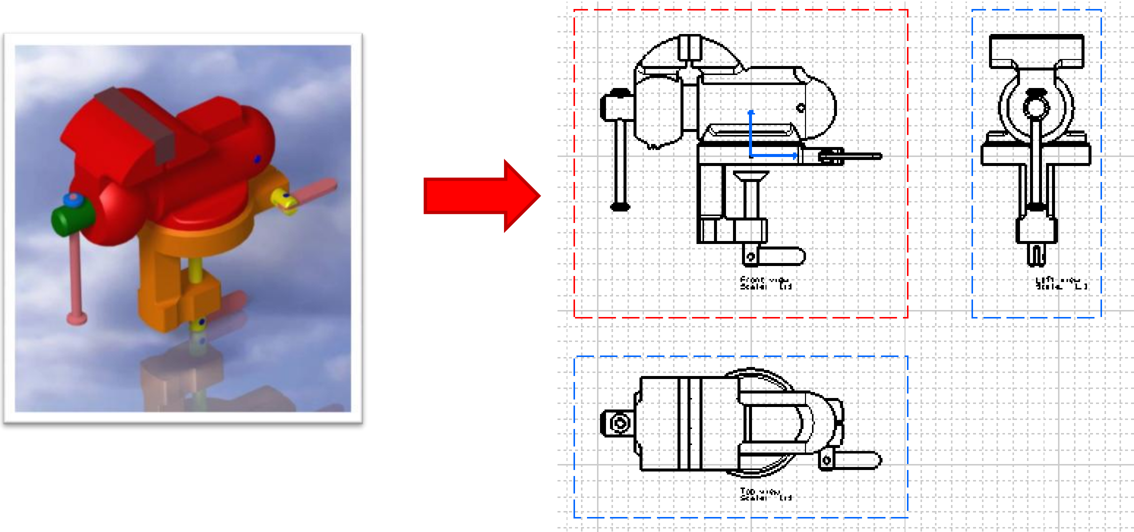
CAD support

- Automatic generating of views from CAD model
- Axonometric view



Follows let's summarize the possibilities of CAD support in the area of create engineering drawing.

The first and the most useful is the automatic generation of views based on CAD model. The designer has to focus on the 3D model only, the right and complete drawing representation is the task of the CAD system. There are no missing lines, no bad projection.

We can generate not only the conventional view, but the axonometric view too, which is very useful to interpret a drawing.

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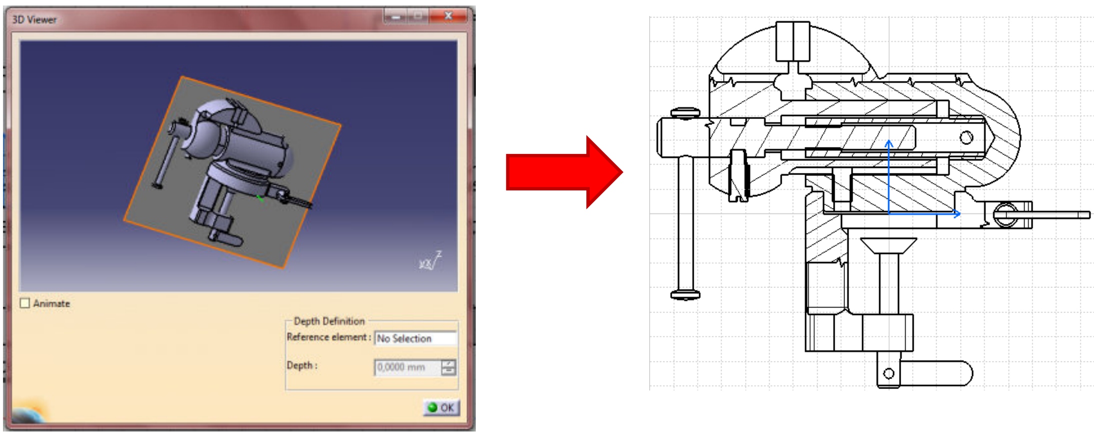
CAD support

- Automatic generation of sections
- Automationg hatching



We can generate sections, and the hatching will generate automatically. Of course it need some adjustment to set the aesthetic hatching angle and density or the non visible or non sectioned parts.

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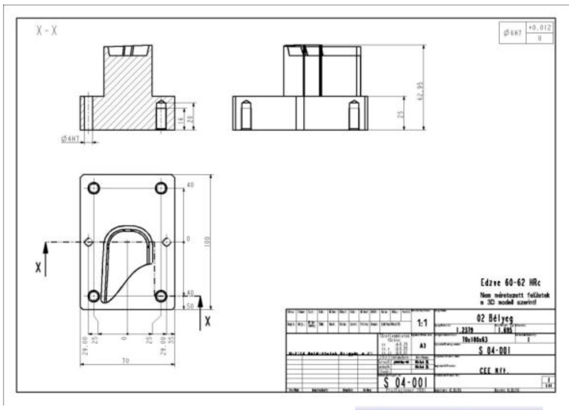
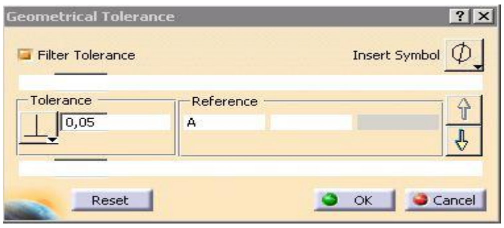
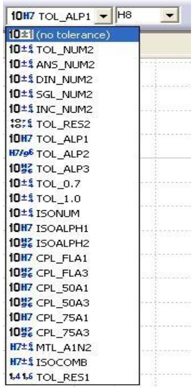
CAD support

- Changeable sheet size
- Predefine formats
- Automatic text box
- Automatic dimensioning
- Tolerance
- Automatic hole position table







We can change the size of the sheet during the drawing, so if the first choice is too small or too big, we can increase or decrease it. The position of the views are changeable too and the projected views will move with the main view.

We can define special sheet formats with textbox and layouts, and the textbox will fill up based on 3D model's parameter table. So the basic data will define only in one place.

The dimensioning needs lot of time in the drawing process, but CAD systems ensure automatic dimensioning. The system will appear the dimensions in the selected views. But the system can draw up only that dimensions, which were defined during the modelling process in the selected view plane. The efficient application of this function needs lot of practice and forecast during the modelling.

The definition of tolerances can be set automatically too. The CAD systems can change the formats of the tolerances, we can define them by ISO marks, or limits or upper and down values etc.

Based on CAD model we can create a hole position table for the production. So the designed should not collect the position of the holes by diameters and lengths, which takes time and the automatic table is associative with the model.

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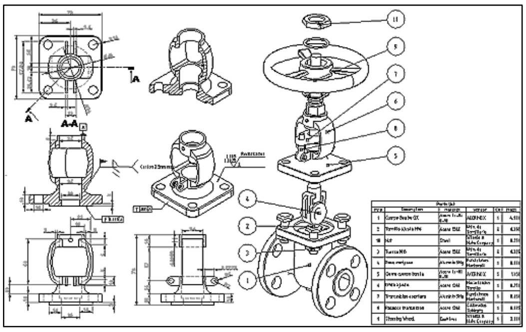
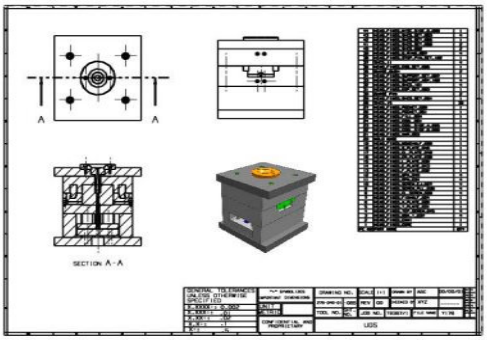
CAD support

- Assembly drawing from assembly model
- Exploded view
- Bill of material and numbering





An assembly drawing can be generated from the assembly model as in case of part model and drawing. The process and the possibilities are same.
The exploded view can be generated from arbitrary view, which makes the understanding of the design easy.
The bill of material contains the list of elements and the essential data of them. It can generated automatically from the assembly model based on data sheet of the model.

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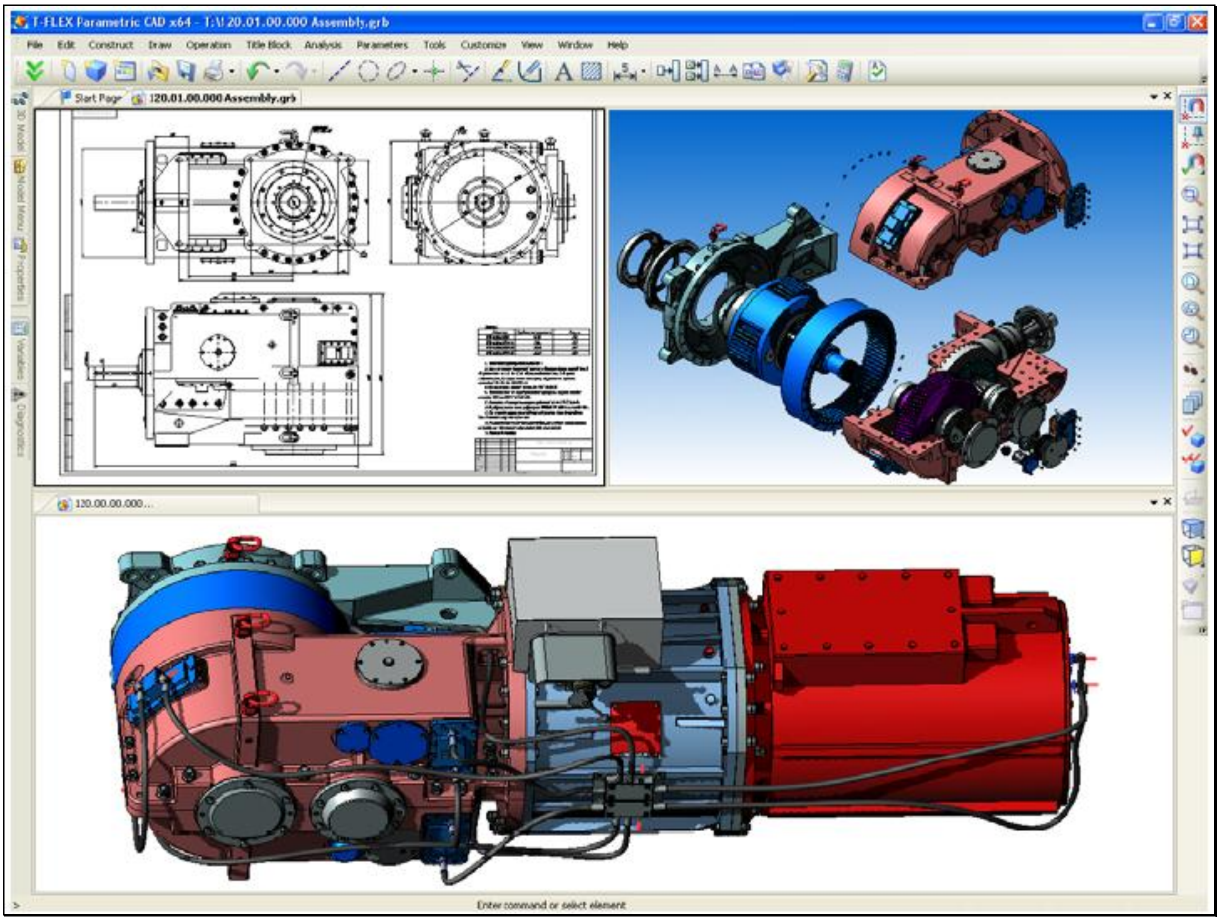

This picture shows the assembly drawing, the exploded view and the assembly model of a complete electric motor and gear box. The different segments of the working place show different level of the information and the interactive model helps to understand the complex layout of the device.

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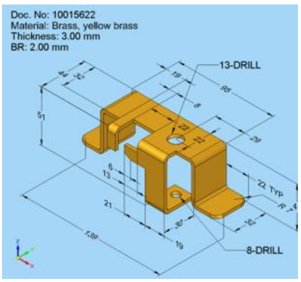
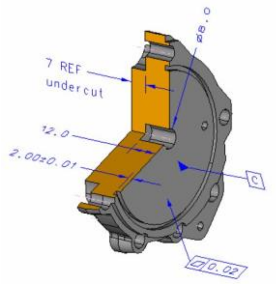
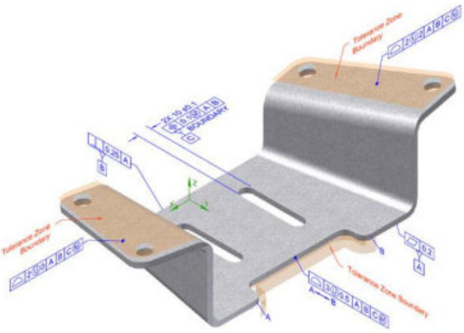
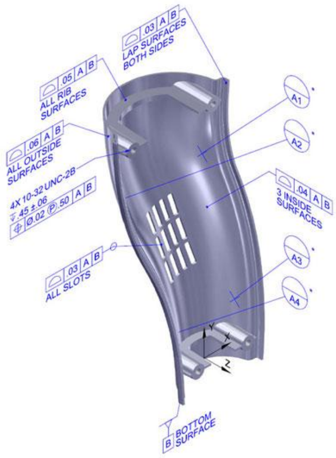
3D drawing

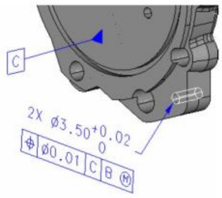
- ASME Y14.41-2003
- No paper document
- Special symbols
- Interactive doc.

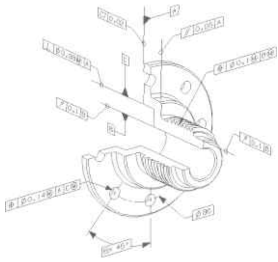












The CAD systems established the capability to create a new form of engineering drawing. The 3D drawing ensure the paperless documentation, so it is exists only electronic format. The idea came from Toyota, and the details are documented in ASME Y14.41-2003 standard. The most significant CAD systems support the application of this standard.
The 3D drawing is an electronic interactive document, the user can be rotate, zoom, set different views and sections.

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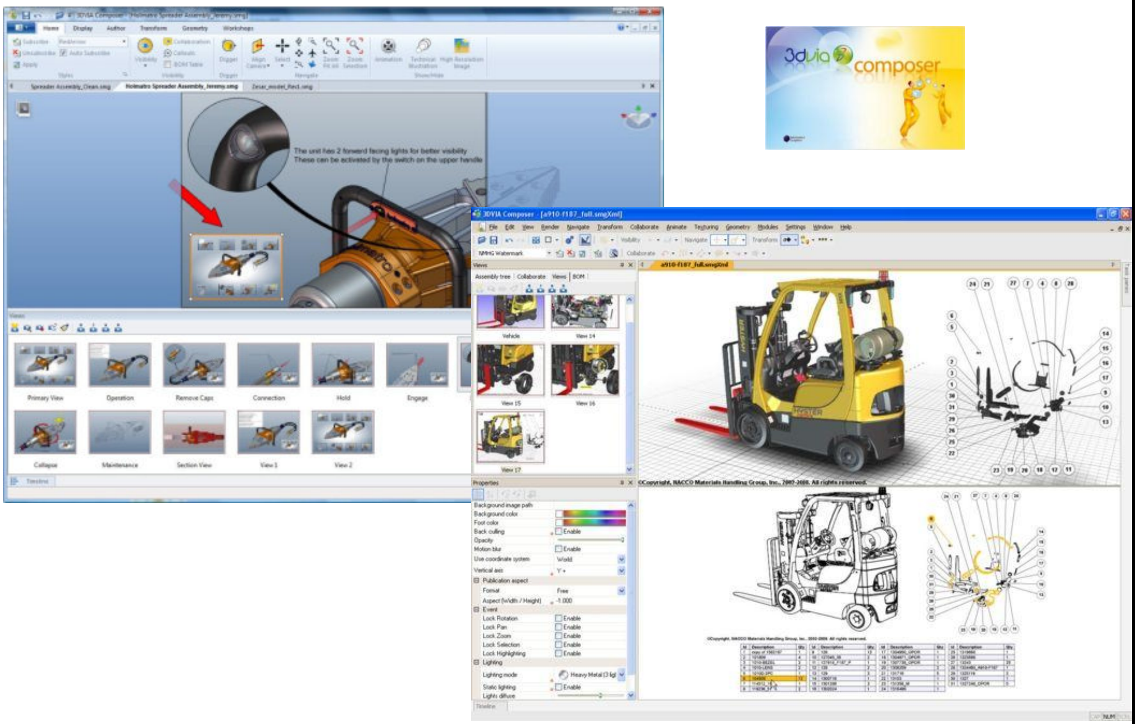
Interactive document



The interactive documenting has an increasing role in the area of engineering collaboration. This tools, like „3D via Composer”, ensure lot of facilities to define different views, exploded views, hide segments, create notes etc.

The generated file doesn't required CAD system to view, only a viewer, and the file size is very small. The user can measure in the model, but cannot modify or copy geometry.

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ENGINEERING DRAWING

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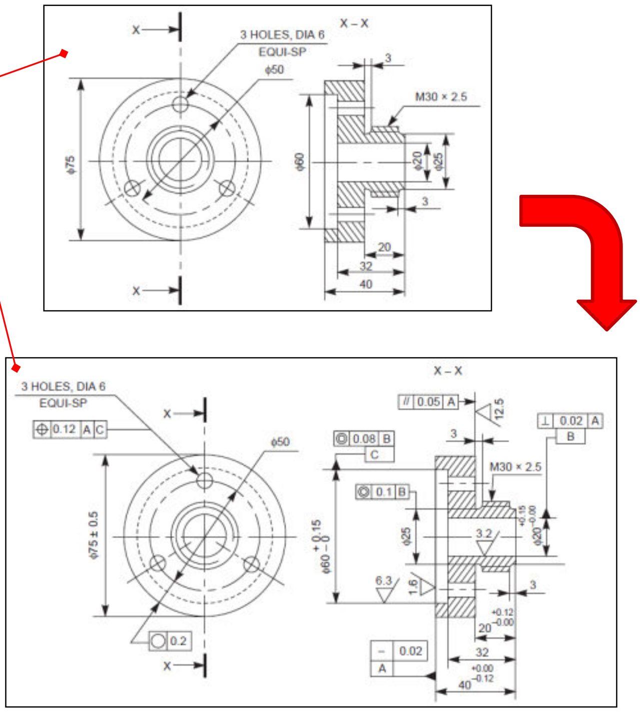
Classification of drawings

- Machine drawing
- Production drawing
- Part drawing



In this section the principle of engineering drawing will summarize. The drawings can be classifyed as machnice drawing, productuon drawing, part drawing, assembly drawing, etc.

Machine drawing is pertaining to machine parts or components. It is presented through a number of orthographic views, so that the size and shape of the component is fully understood. Part drawings and assembly drawings belong to this classification.

A *production drawing*, also referred to as working drawing, should furnish all the dimensions, limits and special finishing processes such as heat treatment, honing, lapping, surface finish, etc., to guide the craftsman on the shop floor in producing the component. The title should also mention the material used for the product, number of parts required for the assembled unit, etc.

Component or *part drawing* is a detailed drawing of a component to facilitate its manufacture. All the principles of orthographic projection and the technique of graphic representation must be followed to communicate the details in a part drawing. A part drawing with production details is rightly called as a production drawing or working drawing.

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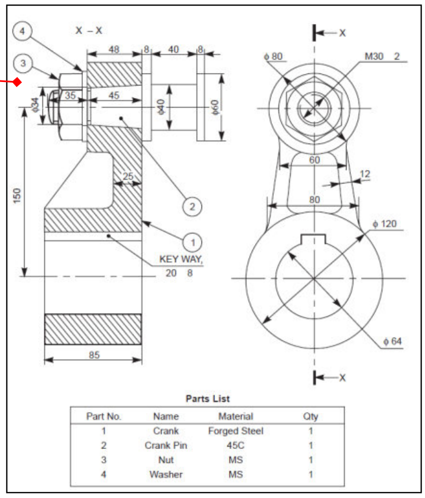
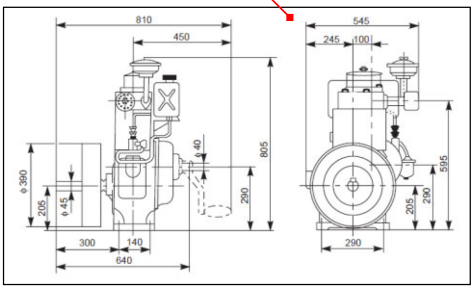
Classification of drawings

- Assembly drawing
  - Design assembly drawing
  - Detailed assembly drawing
  - Sub-assembly drawing
  - Installation assembly drawing
  - Catalogue drawing





Assembly drawing: A drawing that shows the various parts of a machine in their correct working locations is an
assembly drawing. There are several types of such drawings.

Design assembly drawing: When a machine is designed, an assembly drawing or a design layout is first drawn to clearly visualise the performance, shape and clearances of various parts comprising the machine.

Detailed assembly drawing: It is usually made for simple machines, comprising of a relatively smaller number of simple parts. All the dimensions and information necessary for the construction of such parts and for the assembly of the parts are given directly on the assembly drawing. Separate views of specific parts in enlargements, showing the fitting of parts together, may also be drawn in addition to the regular assembly drawing.

Sub-assembly drawing: A sub-assembly drawing is an assembly drawing of a group of related parts, that form a part in a more complicated machine.

Installation assembly drawing: On this drawing, the location and dimensions of few important parts and overall dimensions of the assembled unit are indicated. This drawing

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provides useful information for assembling the machine, as this drawing reveals all parts of a machine in their correct working position.

Catalogue drawing: Special assembly drawings are prepared for company catalogues. These drawings show only the pertinent details and dimensions that would interest the potential buyer.

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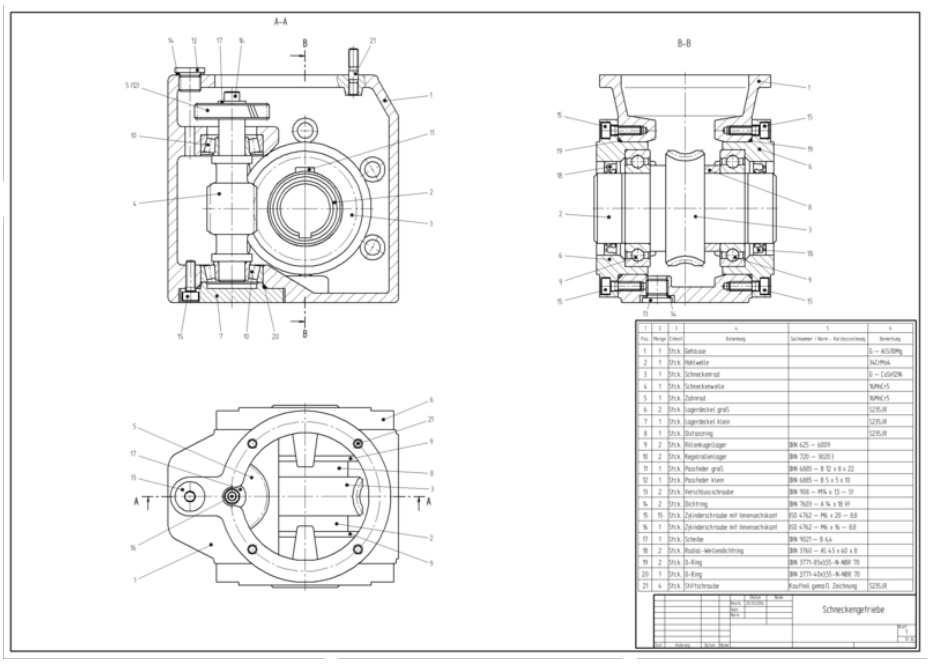
Classification of drawings

- Assembly drawing



The assembly drawing shows the connection of parts. It visualize the layout of the machine, the dimensions of the connection elements. The parts are identified by numbers and the bill of material (BOM) shows the basic data of them.

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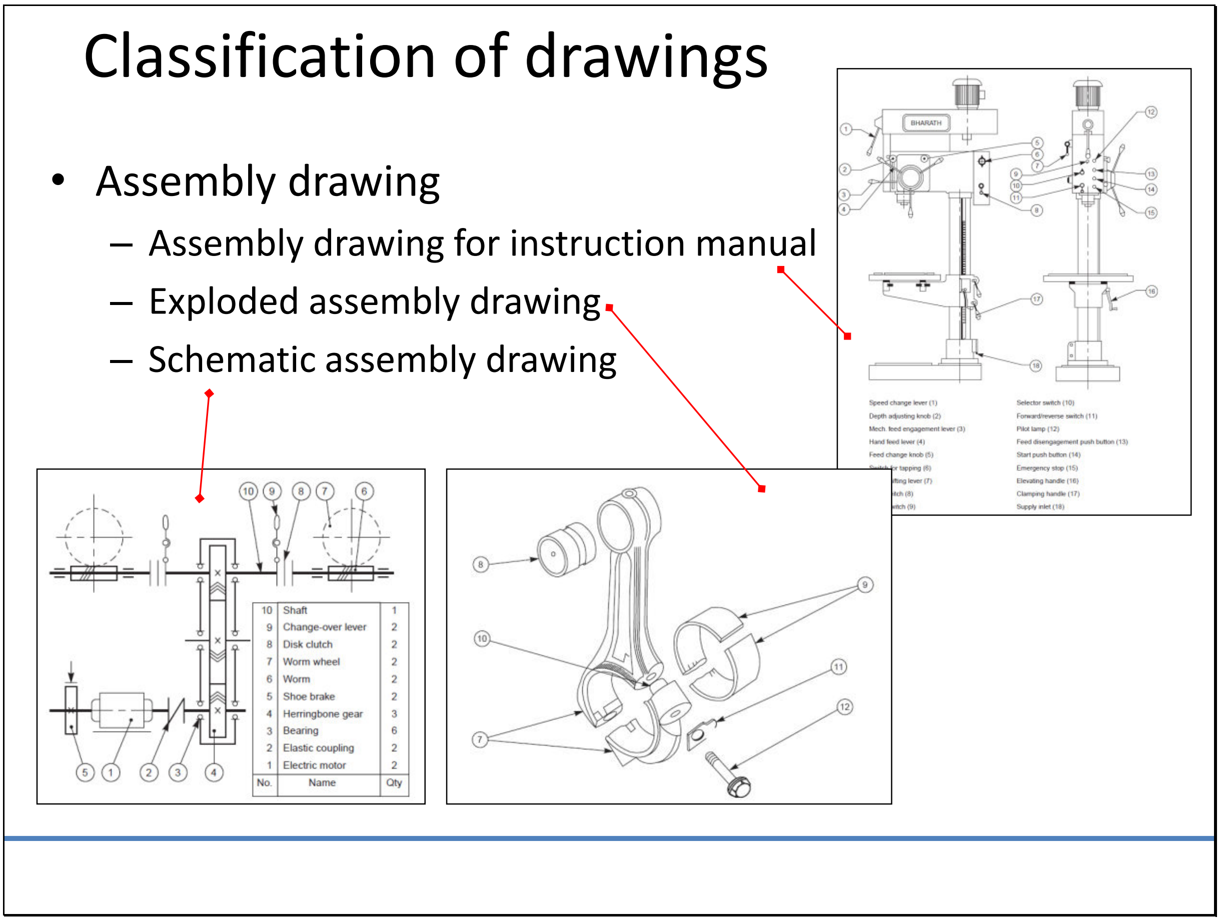

The assembly drawing has some sub-classes:

Assembly drawing for instruction manual: These drawings in the form of assembly drawings, are to be used when a machine, shipped away in assembled condition, is knocked down in order to check all the parts before reassembly and installation elsewhere. These drawings have each component numbered on the job.

Exploded assembly drawing: In some cases, exploded pictorial views are supplied to meet instruction manual requirements. These drawings generally find a place in the parts list section of a company instruction manual.

Schematic assembly drawing: It is very difficult to understand the operating principles of complicated machinery, merely from the assembly drawings. Schematic representation of the unit facilitates easy understanding of its operating principle. It is a simplified illustration of the machine or of a system, replacing all the elements, by their respective conventional representations.

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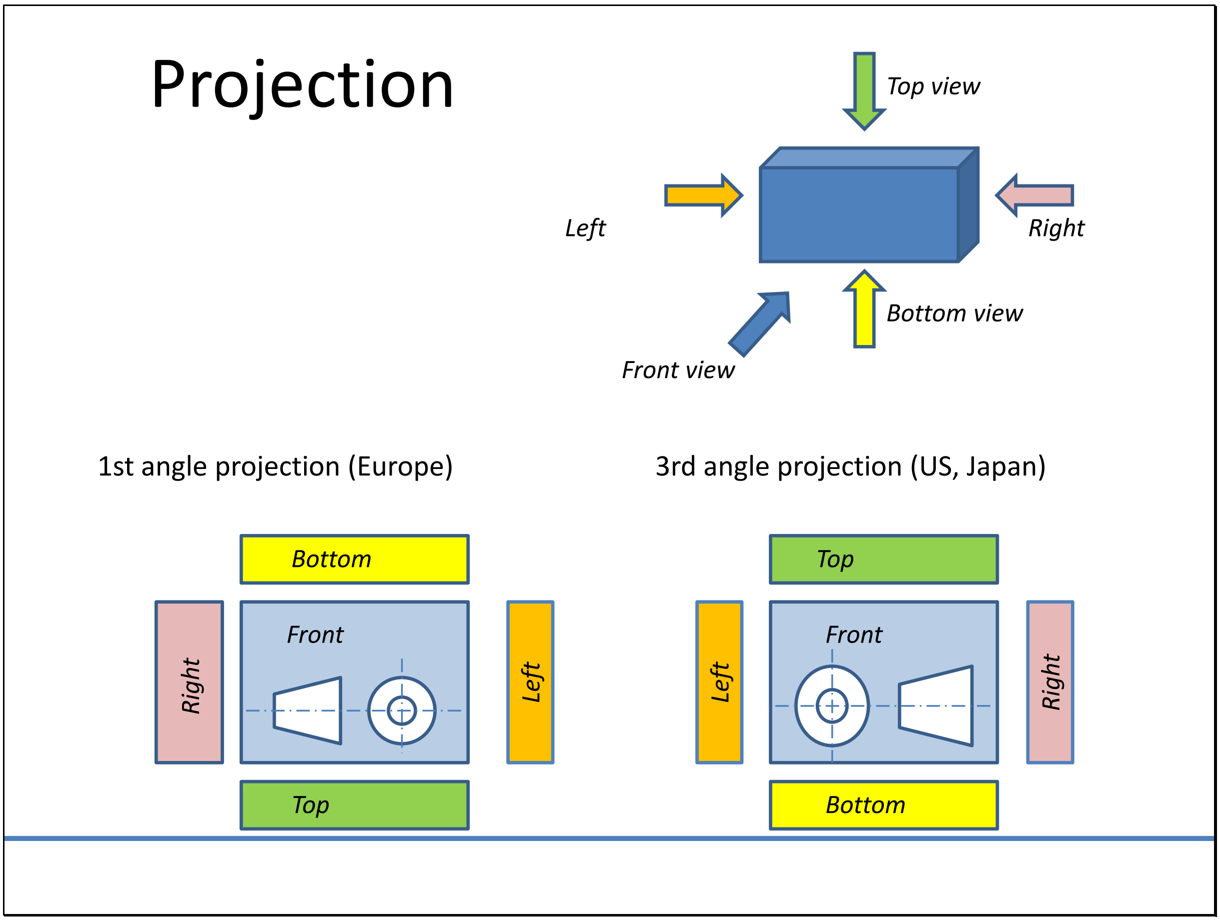

A projection may be obtained by viewing the object from the point of sight and tracing in correct sequence, the points of intersection between the rays of sight and the plane on to which the object is projected. A projection is called orthographic projection when the point of sight is imagined to be located at infinity so that the rays of sight are parallel to each other and intersect the plane of projection at right angle to it.
An object positioned in space may be imagined as surrounded by six mutually perpendicular planes. So, for any object, six different views may be obtained by viewing at it along the six directions, normal to these planes.

Ther are two tipes of projection:

- 1st angle projection,
- 3rd angle projection

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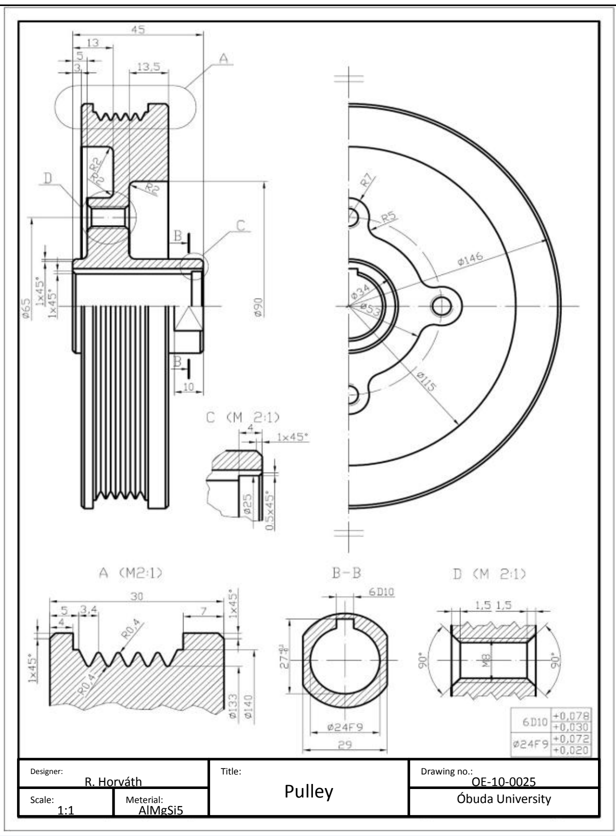

An object in an engineering drawing is visualised by minimum number of views. The front view contains the lot of information.
The size of the sheet is selected to the best filling of it. It won't be empty or crammed.
The scale of the drawing should harmonize with the size of the sheet and visibility.

Some rule about views:

- Minimal number of views
- Clarity
- Visibility
- The front view contains the lot of information.
- An axonometric view support the interpretation.
- Use the whole space.
- Don't be crammed.

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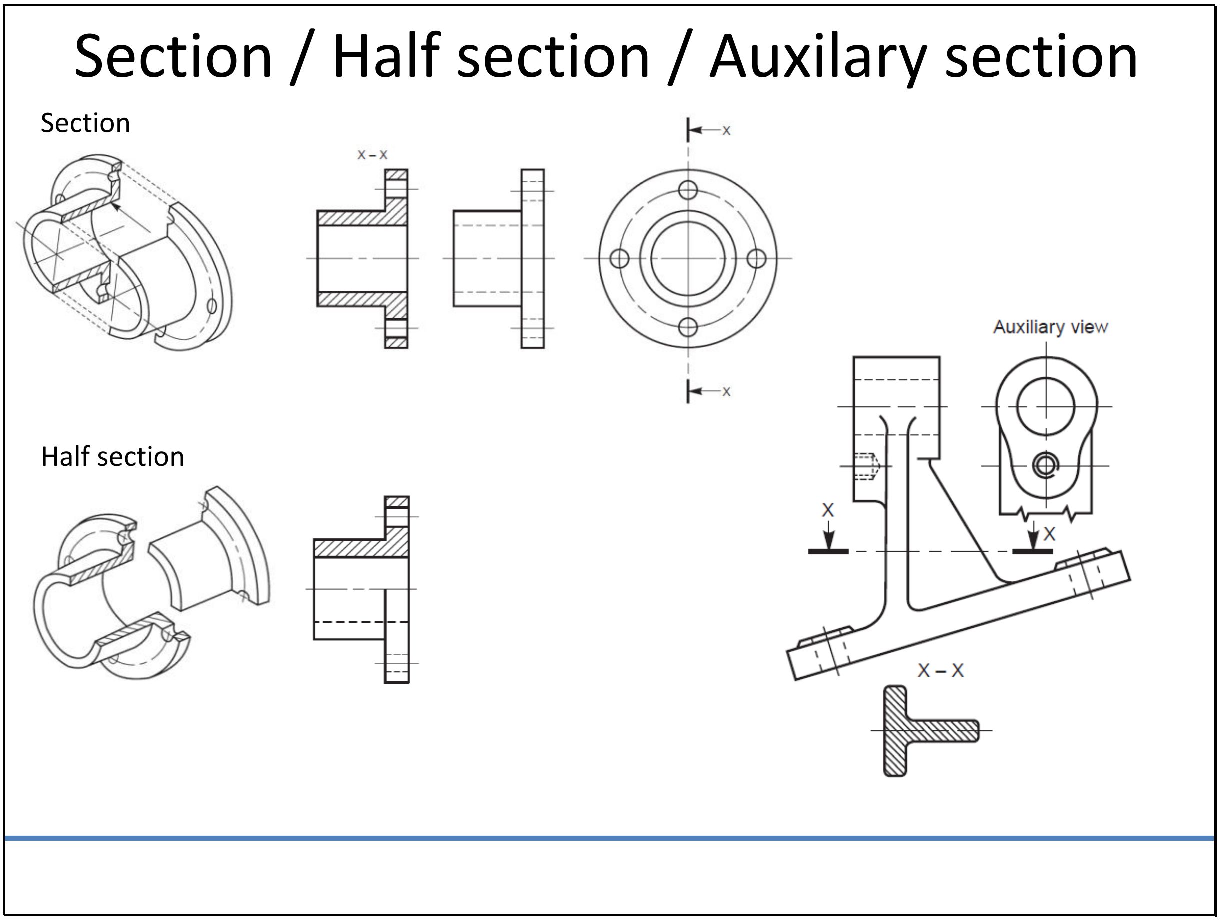

The details of a part or assembly can be presented by sectional views, hlaf sections or auxilary sections.

A *sectional view* obtained by assuming that the object is completely cut by a plane is called a full section or sectional view. The sectioned view provides all the inner details, better than the unsectioned view with dotted lines for inner details. The cutting plane is represented by its trace in the view from the front and the direction of sight to obtain the sectional view is represented by the arrows.

A *half sectional view* is preferred for symmetrical objects. For a half section, the cutting plane removes only one quarter of an object. For a symmetrical object, a half sectional view is used to indicate both interior and exterior details in the same view. A a centre line is used to separate
the halves of the half section.

Auxiliary sections may be used to supplement the principal views used in orthographic projections. A sectional view projected on an auxiliary plane, inclined to the principal planes of projection, shows the cross-sectional shapes of features such as arms, ribs and so on.

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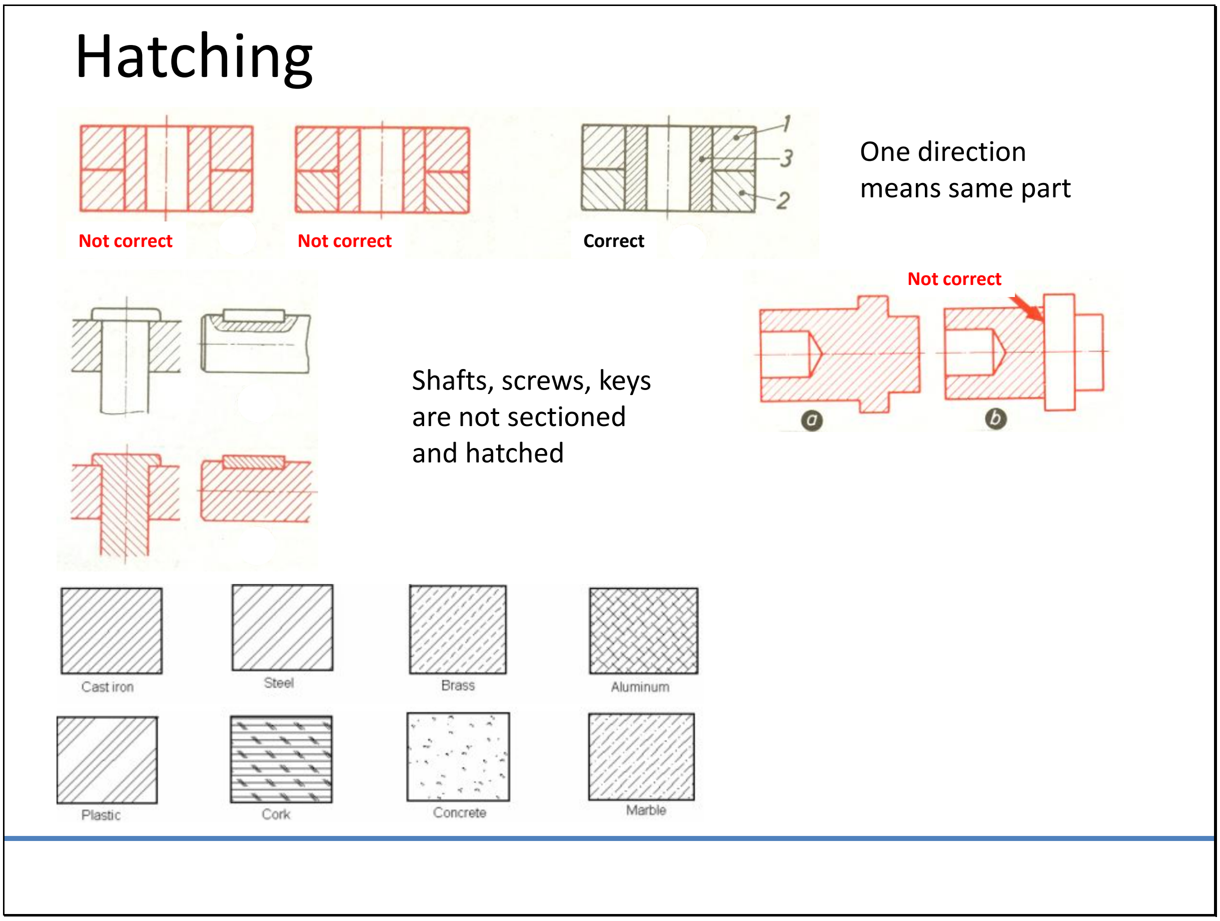

Hatching is generally used to show areas of sections. The simplest form of hatching is generally adequate for the purpose, and may be continuous thin lines at a convenient angle, preferably 45°, to the principal outlines or lines of symmetry of the sections.
Separate areas of a section of the same component shall be hatched in an identical manner. The hatching of adjacent components shall be carried out with different directions or spacing. In case of large areas, the hatching may be limited to a zone, following the contour of the hatched area.
Where sections of the same part in parallel planes are shown side by side, the hatching shall be identical, but may be off-set along the dividing line between the sections. Hatching should be interrupted when it is not possible to place inscriptions outside the hatched area.

Some rule about sectional views:

- The one direction of hatching means same part
- The shafts, screws, keys are not sectioned and hatched
- The boundary of the section be a separated line

The pattern of the hatching characterizes the type of the material:

- Metal: 45° continuous line
- Plastic, rubber: 3 lines in 45°

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- Glass or transparent materials: 45° short line with end section
- Liquid: short horizontal lines
- Wood: wood similar pattern
- Concrete / cement / granulated material: random point cloud

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Text information

- Title block (title, ID number, designer's name, material, scale, weight, overall size, sheet size etc.)
- CAD file name
- General fillet and chamfer
- General tolerances
- Surface roughness
- Wall thichness
- Engraving text in the surface
- Testing requirements
- Modification data

The title block should lie within the drawing space such that, the location of it, containing the identification of the drawing, is at the bottom right hand corner. This must be followed, both for sheets positioned horizontally or vertically. The direction of viewing of the title block should correspond in general with that of the drawing. The title block can have a maximum length of 170 mm.

A typical title block, providing the following information:

- Title of the drawing
- Sheet number
- Scale
- Symbol, denoting the method of projection
- Name of the firm
- Initials of staff drawn, checked and approved.

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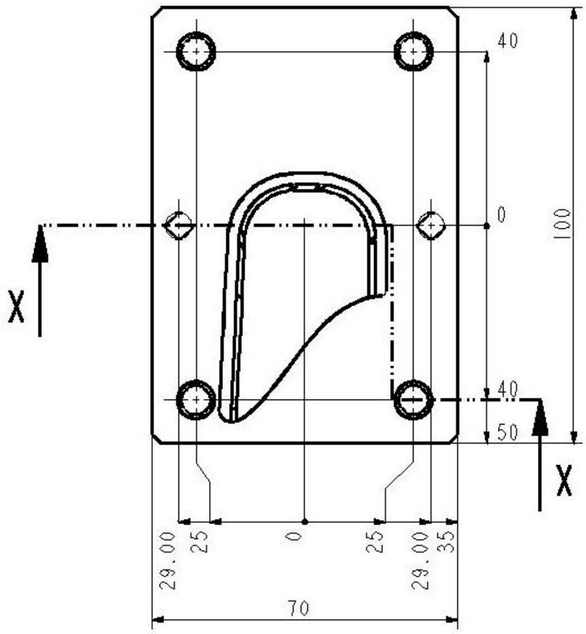
Datum elements for dimensioning



Design base = Manufacturing base → Main base

The key element of the dimensioning of a machine part is the datum element. The datum element is a geometric feature of the part, which is the reference of the identify of other geometric features.

The datum element can be

- Design datum, which is defined by the designer based on the functional viewpoints, or
- Manufacturing datum, which consider the manufacturing viewpoints.

If these two type of datum are same, it call main datum.

On other classification the datum feature can be

- Real geometric element, like the side of the part, or
- Theoretical element, like the centre of a hole.

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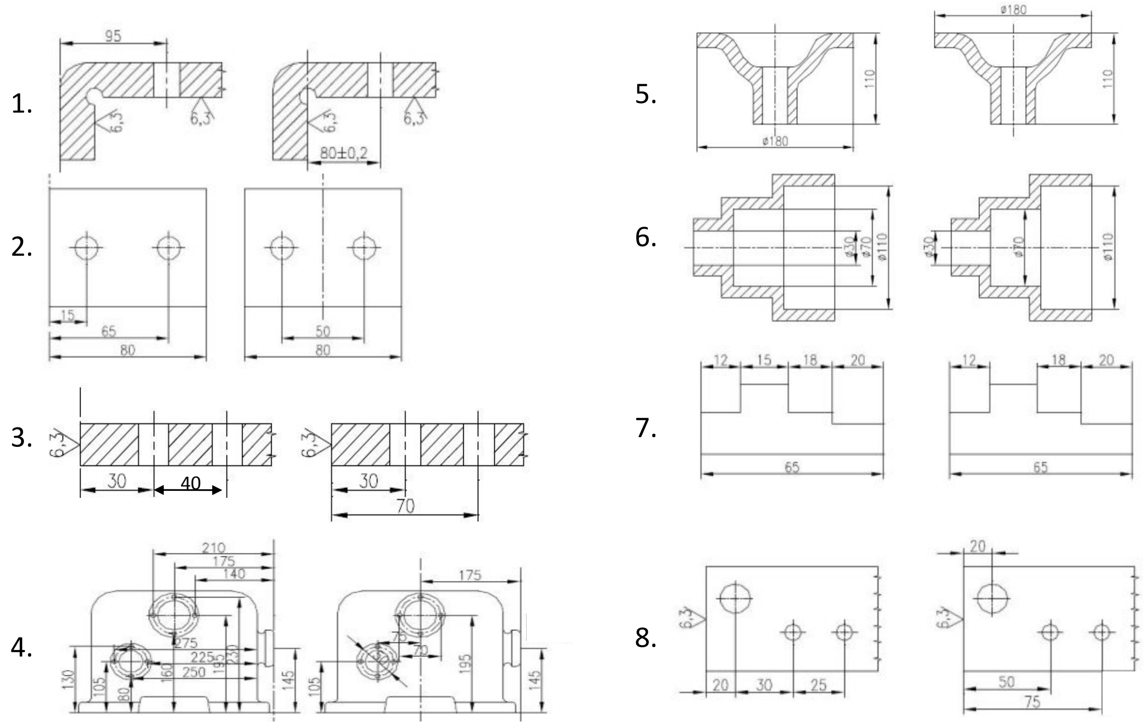
The rule of dimensioning



When the dimension system is defined the designer have to follow some roles and directives.

Some important directives of dimensioning are follows:

- Clarity
- Readability
- Aesthetic
- Same feature in same view
- Everything only once
- Design, manufacturing and measure viewpoint in same time

The pictures show examples:

- Use the manufactured surface, as base.
- Use symmetric dimensioning, if the part is symmetric
- Avoid the chain dimensioning cause the toleranceing
- Use local base elements
- Put the dimension line close to the surface
- Use as short projection line as possible
- Avoid redundant dimensioning
- Avoid the chain dimensioning cause the tolerancing

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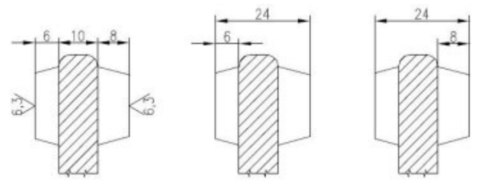
The rules of dimensioning

9.



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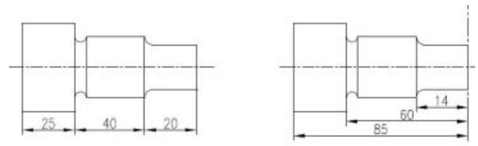

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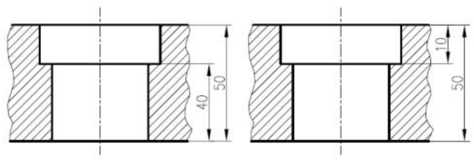

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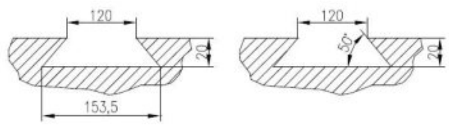

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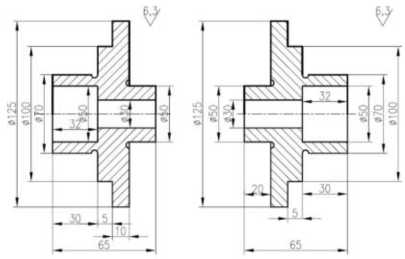

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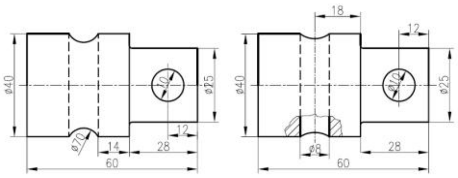

- Define the dimension of the machined surfaces
- Take consider the method of machining
- Take consider the method of machining
- Take consider how can you measure the defined dimensions
- Group the dimensions according to the machining setups
- Group the dimensions according to the machining setups

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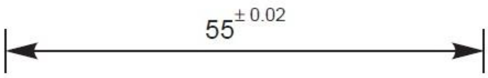
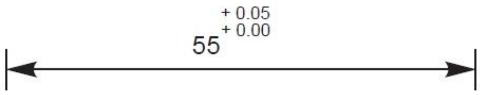
Tolerances

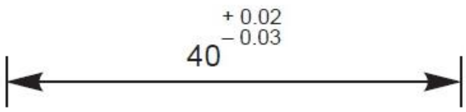
- **Tolerance = the admitted error**
- Size
  - General tolerance
  - With limits
  - IT fits
- Form
- Position



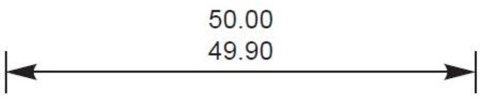
Bilateral tolerance of equal variation



Unilateral tolerance with zero variation in one direction



Bilateral tolerance of unequal variation



Maximum and minimum size directly indicated

Because of the manufacturing inaccuracies every dimensions have tolerances. The tolerance is the admitted error. We can talk about size, form and position tolerances.

The size tolerances can be defined by standard, when no visible tolerance on de dimension, and the value of it is defined by tables. We can add the limits of the differences by numbers of IT classification.

The form and position of the differences can be the next:

- Bilateral tolerance of equal variation
- Unilateral tolerance with zero variation in one direction
- Bilateral tolerance of unequal variation
- Maximum and minimum size directly indicated

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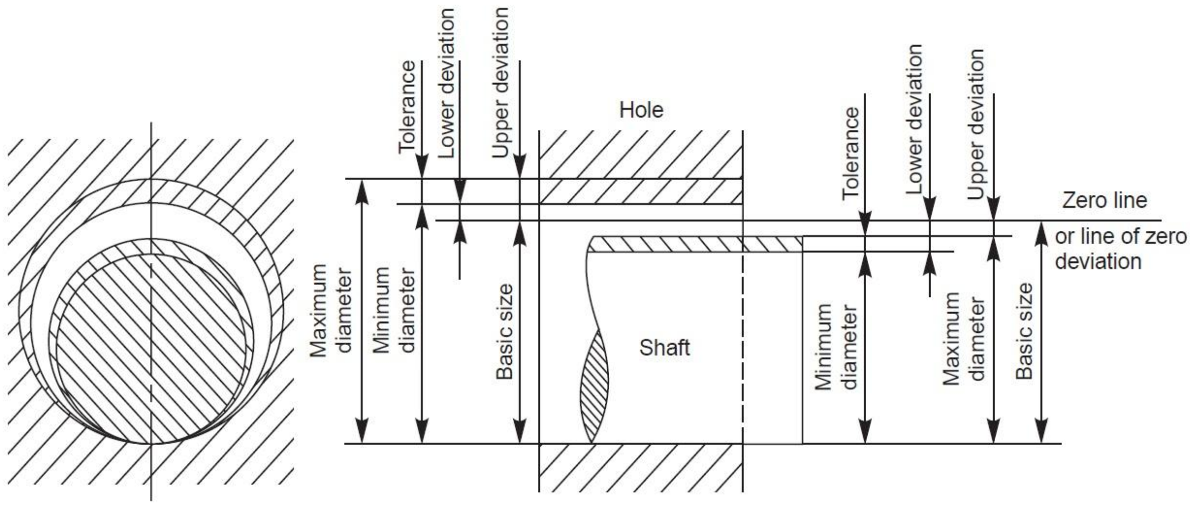

The figure shows the basic term of the size deviations and tolerances.
In case of hole and shaft, the tolerances will identify the fit of these two elements.

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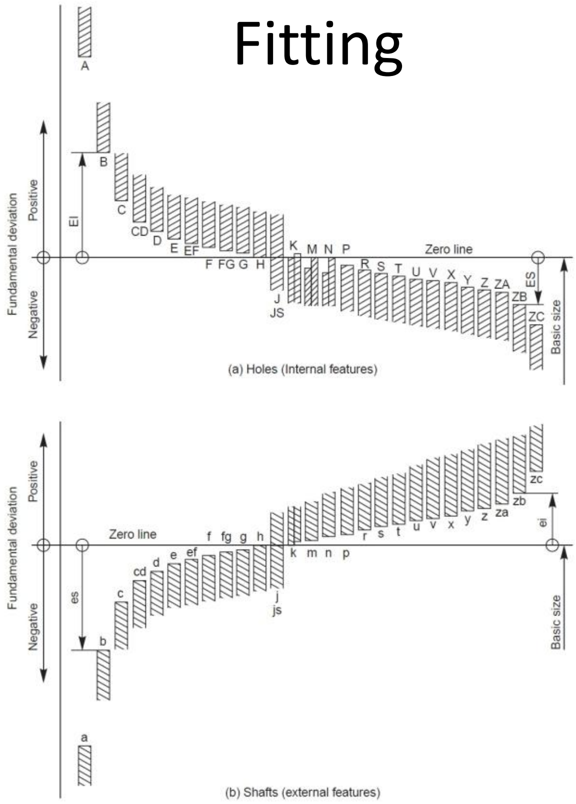
Fitting



The ISO fitting definition makes easy to define the relation of shaft and hole.

The ISO mark, like Ø120H8, shows

- The type of the object: the capital letter (H) marks the hole, the small letter (h) marks the shaft.
- The position of the tolerance zone (see the picture), and
- The wide of the tolerance zone.

The recommended pairs of tolerances in case of different applications are listed in engineer handbooks.

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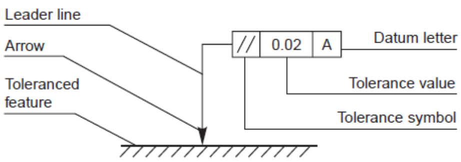
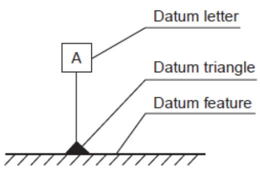
Form, shape and position tolerances





Characteristics to be toleranced		*Symbols*
Form of single features	Straightness	⏤
	Flatness	▱
	Circularity (roundness)	○
	Cylindricity	⌭
	Profile of any line	⌒
	Profile of any surface	⌓
Orientation of related features	Parallelism	//
	Perpendicularity (squareness)	⊥
	Angularity	∠
Position of related features	Position	⌖
	Concentricity and coaxiality	◎
	Symmetry	⌯
	Run-out	↗

The *form variation* is a variation of the actual condition of a form feature (surface, line) from geometrically ideal form.

The *position variation* is a variation of the actual position of the form feature from the geometrically ideal position, with reference to another form (datum) feature.

Geometrical tolerance is defined as the maximum permissible overall variation of form or position of a feature.
Geometrical tolerances are used,

- to specify the required accuracy in controlling the form of a feature,
- to ensure correct functional positioning of the feature,
- to ensure the interchangeability of components, and
- to facilitate the assembly of mating components.

The tolerance of form can be

- Straightness
- Flatness
- Roundness
- Cylindricity

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- Profile of any line
- Profile of any surface

The tolerance of position can be

- Parallelism
- Perpendicularity
- Angularity
- Position
- Concentricity and coaxiality
- Symmetry
- Run-out

In case of tolerance of position, a datum feature has to defined. The value of the tolerances is added in millimetre.

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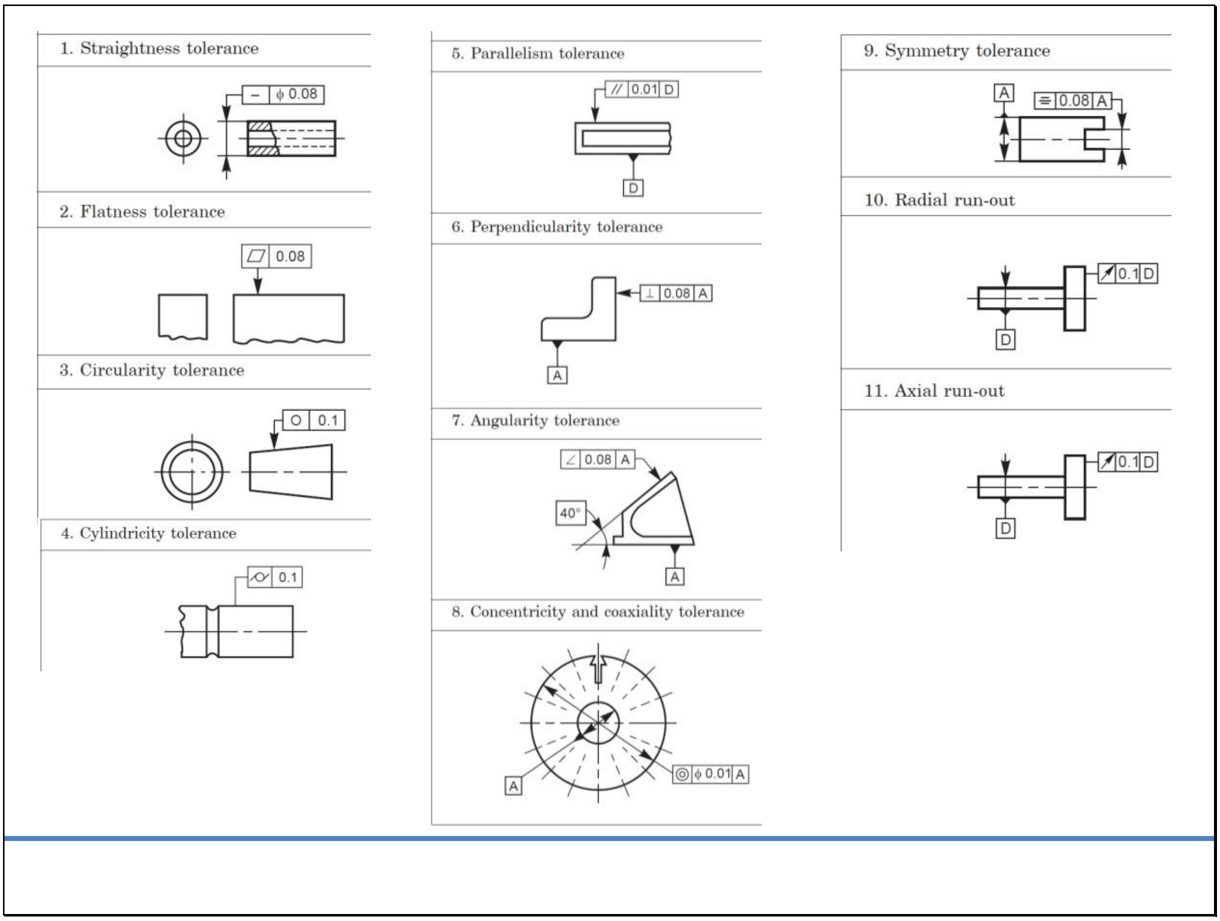

The pictures show examples to application of these tolerances.

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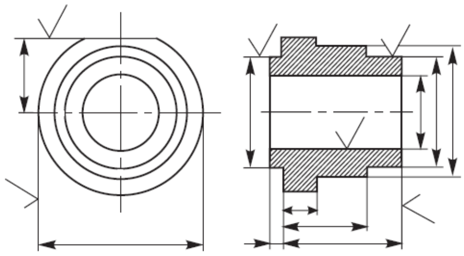
Surface roughness



It is not possible to achieve in practice, a geometrically ideal surface of a component and hence, production drawings of components must also contain information about the permissible surface conditions. Machine components which have undergone machining operation, when inspected under magnification, will have some minute irregularities. The actual surface condition will depend upon the finishing process adopted.

The properties and performance of machine components are affected by the degree of roughness of the various surfaces. The higher the smoothness of the surface, the better is the fatigue strength and corrosion resistance. Friction between mating parts is also reduced due to better surface finish.

A surface texture specified,

- As may be obtained by any production method.
- As must be obtained by removal of material by machining.
- As must be obtained without removal of material.

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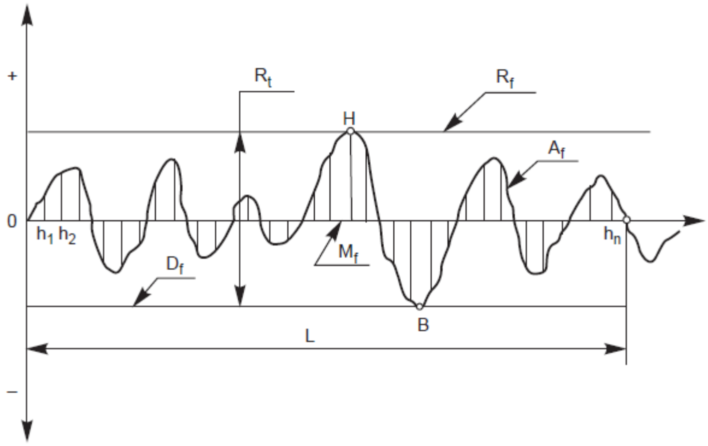
Surface roughness



$$R_a = 1/L \int_{x=0}^{x=L} |h_i| dx$$

The surface roughness is evaluated by the height, R_t and mean roughness index R_a of the micro-irregularities. Following are the definitions of the terms indicated in figure.

A_f - It is the profile of the actual surface obtained by finishing operation.

R_f - It is the profile to which the irregularities of the surface are referred to. It passes through the

highest point of the actual profile.

D_f - It is the profile, parallel to the reference profile. It passes through the lowest point B of the

actual profile.

M_f - It is that profile, within the sampling length chosen (L), such that the sum of the material filled

areas enclosed above it by the actual profile is equal to the sum of the material-void areas enclosed below it by the profile.

R_t - It is the distance from the datum profile to the reference profile.

R_a - It is the arithmetic mean of the absolute values of the heights hi between the actual and mean

profiles.

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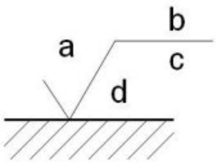
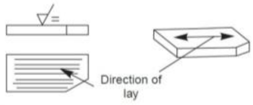
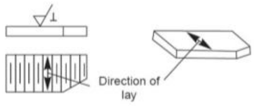
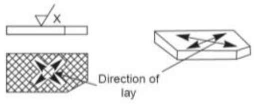
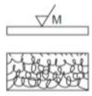
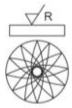
Surface roughness



a – R_a in µm
b - manufacturing technology
c – sampling lenght
d – directions of lay (see table)

Symbol	Interpretation	
=	Parallel to the plane of projection of the view in which the symbol is used	
⊥	Perpendicular to the plane of projection of the view in which the symbol is used	
X	Crossed in two slant directions relative to the plane of projection of the view in which the symbol is used	
M	Multi-directional	
C	Approximately circular, relative to the centre of the surface to which the symbol is applied	
R	Approximately radial, relative to the centre of the surface to which the symbol is applied	

The value of the surface roughness in general mean Ra, but we have to write it before the value, which is in micron.
The direction of the lays, which characterise the surface optical texture can be marked, as the table shows. There is very important to synchronise it to the manufacturing technology.

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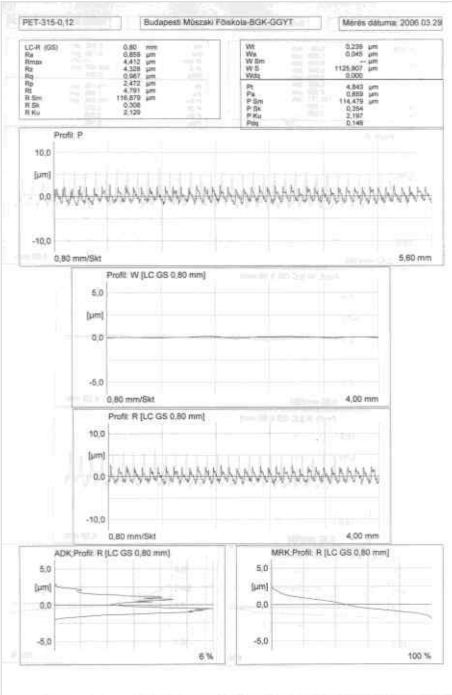
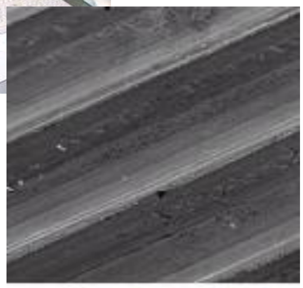
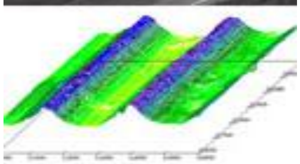
Other parameters







Over the conventional surface roughness parameters we can characterized the surface quality by several other roughness and wave parameters, and based on scanned area we can define several 3D surface roughness parameters.

These parameters are good for compare or classify surfaces, but generally we can’t use them in design phase, because the connection between the surface roughness parameters and the manufacturing process parameters is not known detail.

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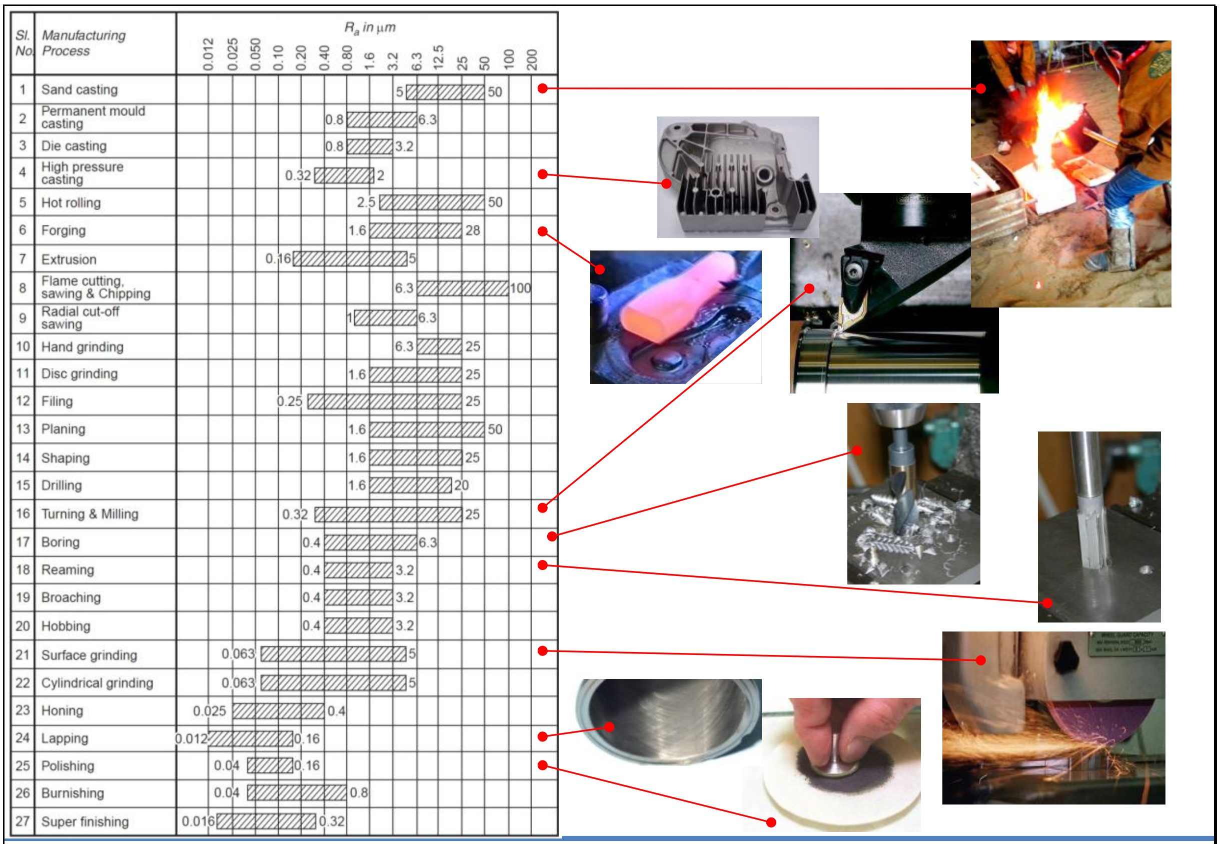

The different production technologies result is different surface quality, so the designer should be consider the manufacturing viewpoints during the identification of required surface quality. The surface quality can be modified by adjust the parameters of the technology, but the changing possibilities are very weak.

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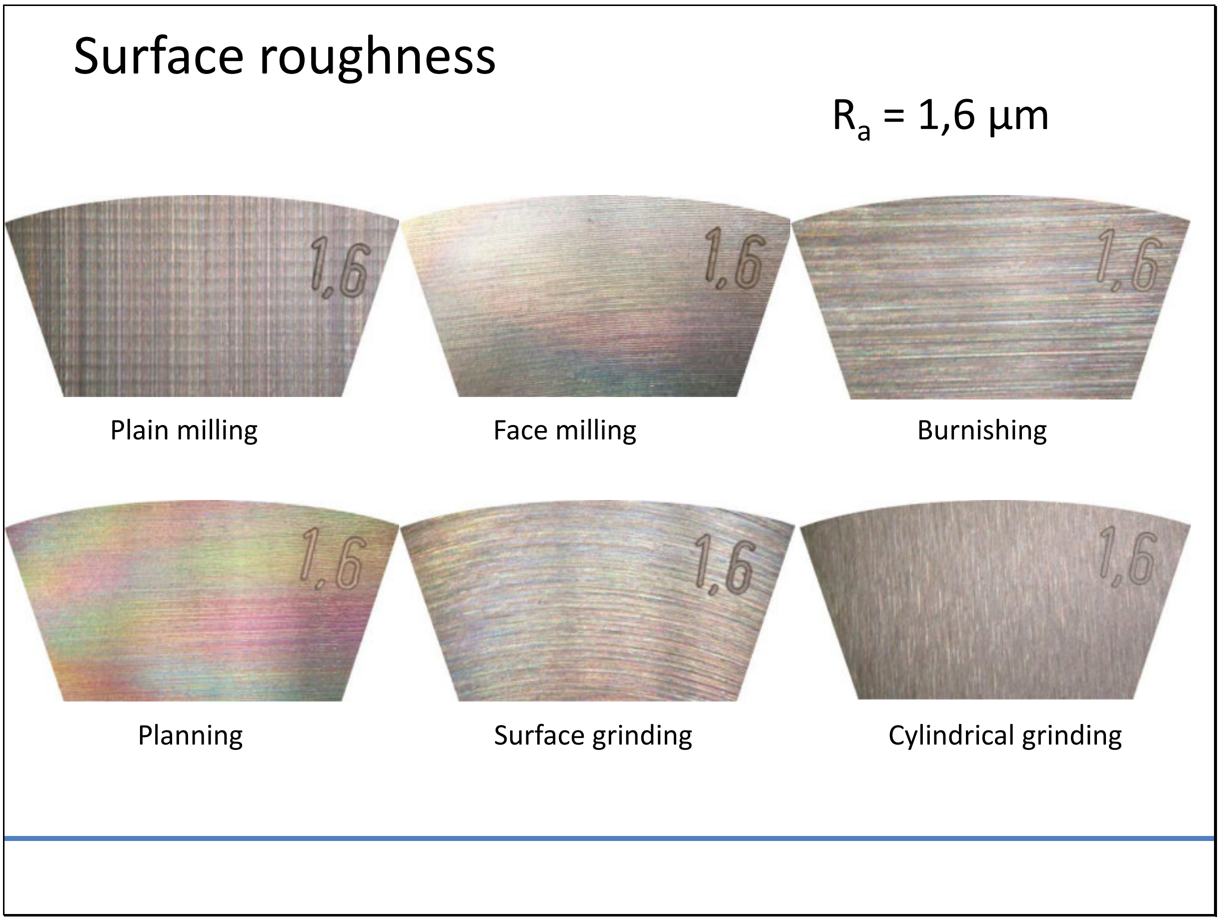

This picture shows six test parts, where the Ra parameter is same, but cause of different machining process the surface textures are different. Sometimes the aesthetic viewpoint is so important as the numeric value of the Ra parameter.

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CAD Book

11. CAD Numerical Methods

Author: Dr. István Oldal
oldal.istvan@gek.szie.hu

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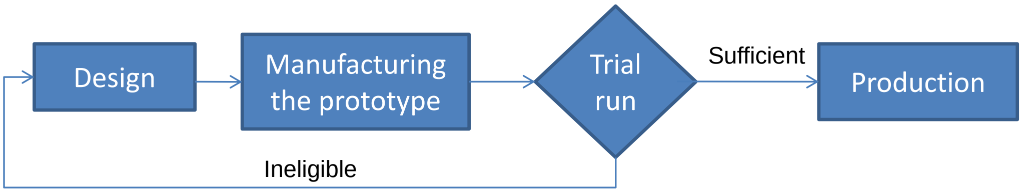
Finite element method in engineering practice



Model of classical production

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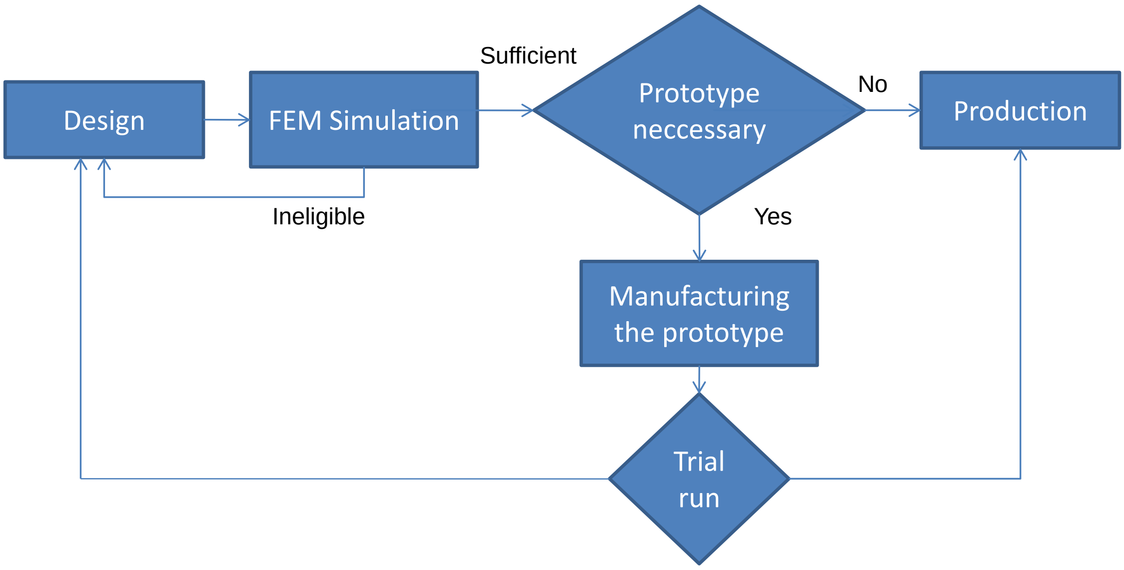
Finite element aided model of production



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Equations used in FEM

- Equilibrium equations
- Geometric equations
- Constitution (material) equations

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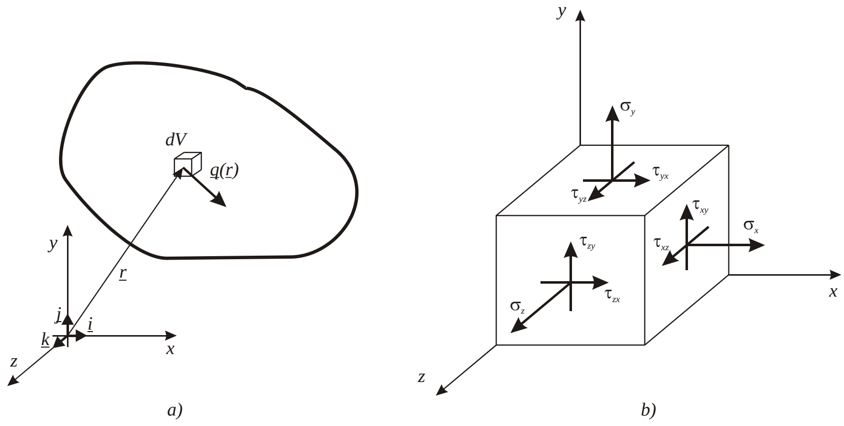
Equilibrium equations

$\underline{q}(\underline{r})$ - distributed force

$\underline{\underline{\sigma}}(\underline{r})$ - stress field tensor



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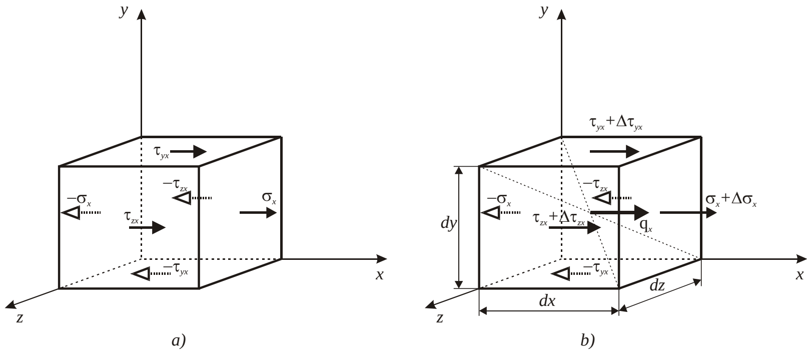
Equilibrium equations

$$\frac{\sigma_x + \Delta\sigma_x}{dx} - \frac{\sigma_x}{dx} + \frac{\tau_{zx} + \Delta\tau_{zx}}{dz} - \frac{\tau_{zx}}{dz} + \frac{\tau_{yx} + \Delta\tau_{yx}}{dy} - \frac{\tau_{yx}}{dy} + q_x = 0$$



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Equilibrium equations

$$d\underline{F} = \underline{q}dV$$

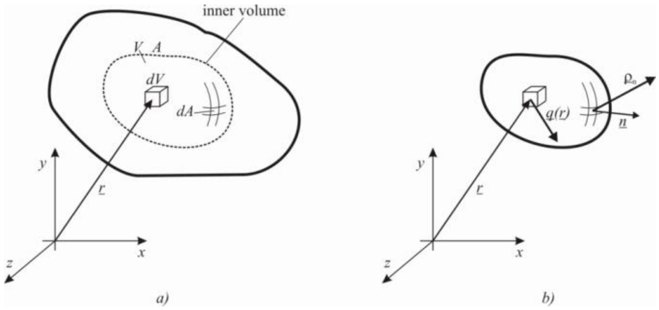
$$d\underline{F} = \underline{\rho}_n dA = \underline{\underline{\sigma}} \cdot \underline{n}dA$$

$$\int_A \underline{\underline{\sigma}} \cdot \underline{n}dA = \int_V \underline{\underline{\sigma}} \cdot \nabla dV$$

$$0 = \int_V \underline{q}dV + \int_V \underline{\underline{\sigma}} \cdot \nabla dV$$

$$0 = \int_V \left(\underline{q} + \underline{\underline{\sigma}} \cdot \nabla\right) dV$$

$$\underline{\underline{\sigma}} \cdot \nabla + \underline{q} = 0$$



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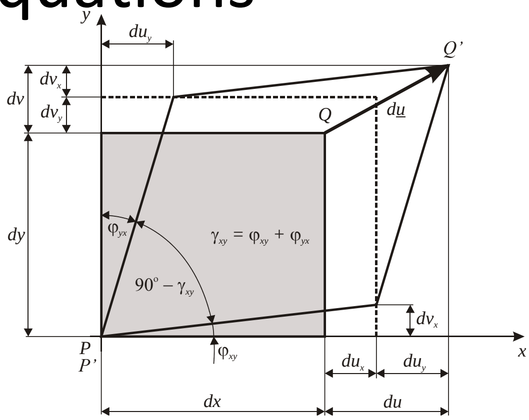
Geometric equations

Deformation:



$$d\underline{u} = du \cdot \underline{i} + dv \cdot \underline{j} + dw \cdot \underline{k}$$

$$\frac{\partial u}{\partial x} = \frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial x} = \frac{du_x}{dx} + 0$$

$$\frac{\partial u}{\partial y} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial y} = 0 + \frac{du_y}{dy}$$

Angle torsion:

$$\gamma_{xy} = \varphi_{xy} + \varphi_{yx} = \arctan\frac{dv_x}{dx} + \arctan\frac{du_y}{dy} \approx \frac{dv_x}{dx} + \frac{du_y}{dy}$$

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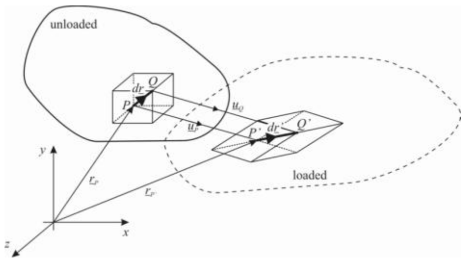
Geometric equations

$$\varepsilon_x = \frac{\partial u}{\partial x}$$

$$\varepsilon_y = \frac{\partial v}{\partial y}$$

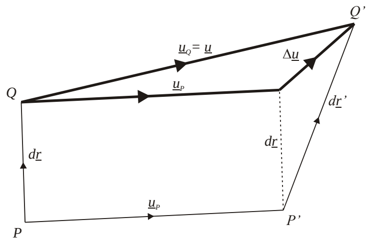
$$\varepsilon_z = \frac{\partial w}{\partial z}$$



$$\gamma_{xy} = \gamma_{yx} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$$

$$\gamma_{yz} = \gamma_{zy} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}$$

$$\gamma_{xz} = \gamma_{zx} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}$$



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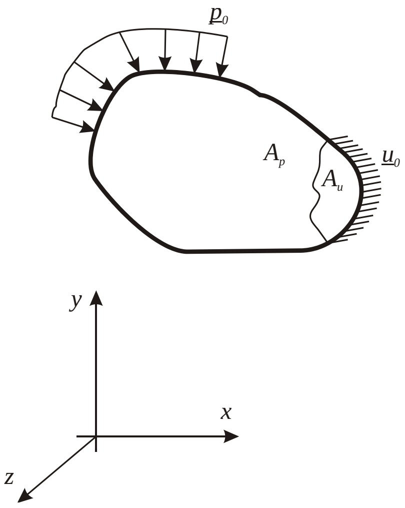
Boundary conditions



- Kinematic boundary conditions

$$\underline{u} = \underline{u}_0$$

- Dynamic boundary conditions

$$\underline{p} = \underline{p}_0 \quad \text{or} \quad \underline{\underline{\sigma}} \cdot \underline{n} = \underline{p}_0$$

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Boundary element method

$$\underline{\underline{\sigma}} \cdot \nabla + \underline{q} = 0$$ - Equilibrium equation

$$\underline{\underline{\varepsilon}} = \frac{1}{2}\left(\underline{u} \circ \nabla + \nabla \circ \underline{u}\right)$$ - Geometric equation

$$\underline{\underline{\sigma}} = 2G\left(\underline{\underline{\varepsilon}} + \frac{\nu}{1-2\nu}\varepsilon_1 \underline{\underline{E}}\right)$$ - Constitutive equation

$$\underline{u}\big|_{A_u} = \underline{u}_0$$ - Kinematic boundary conditions

$$\underline{\underline{\sigma}} \cdot \underline{n}\big|_{A_p} = \underline{p}_0$$ - Dynamic boundary conditions

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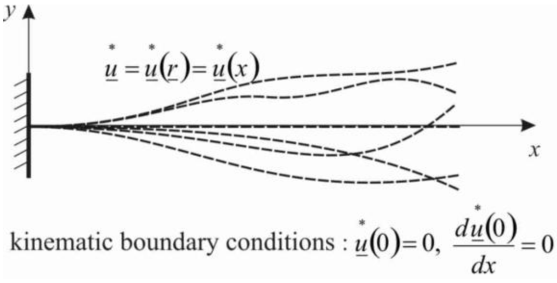
Kinematically admissible displacement field

$$\overset{*}{\underline{\underline{\varepsilon}}} = \frac{1}{2}\left(\overset{*}{\underline{u}} \circ \nabla + \nabla \circ \overset{*}{\underline{u}}\right)$$



$$\overset{*}{\underline{\underline{\sigma}}} = 2G\left(\overset{*}{\underline{\underline{\varepsilon}}} + \frac{\nu}{1-2\nu}\overset{*}{\varepsilon}_1 \underline{\underline{E}}\right)$$

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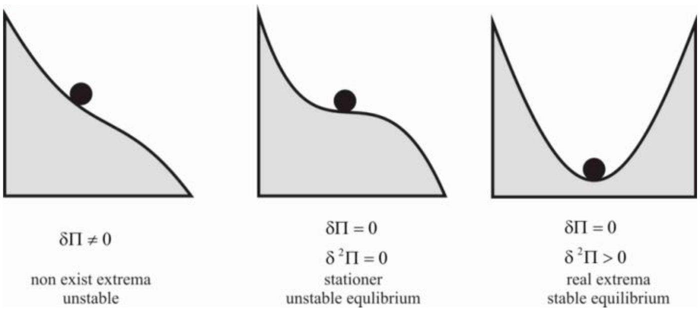
Principle of Lagrange variation

- Minimum total potential energy

$$\Pi[\underline{u}] = U[\underline{u}] - W_k[\underline{u}]$$



Kinematic boundary:

$$\delta \underline{u}\big|_{A_u} = 0$$

The condition of the extrema is: $\delta\Pi = 0$ $\delta\Pi = \delta U - \delta W_k = 0$

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Statically admissible stress field

- Stress field is admissible if:
  - Satisfies the dynamic boundary conditions:

$$\underline{\underline{\sigma}} \cdot \underline{n}\Big|_{A_p} = \underline{p}_0$$

  - Satisfies the equilibrium equations:

$$\underline{\underline{\sigma}} \cdot \nabla + \underline{q} = \underline{0}$$

$$\underline{\underline{\varepsilon}} = \frac{1}{2G}\left(\underline{\underline{\sigma}} - \frac{\nu}{1+\nu}\sigma_1 \underline{\underline{E}}\right) \longrightarrow$$ Deformation field

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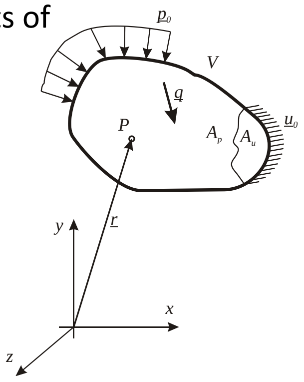
Elasticity problem

Given data:

- The geometry of the body
- The material constants of the body
- Loads
- Constraints



Solution:

- dividing to elements
- elements connects by the nodes
- stress- and deformation field can be derived
- nodal displacement field

Demanded functions: $\underline{u}(\underline{r})$ $\underline{\varepsilon}(\underline{r})$ $\underline{\sigma}(\underline{r})$

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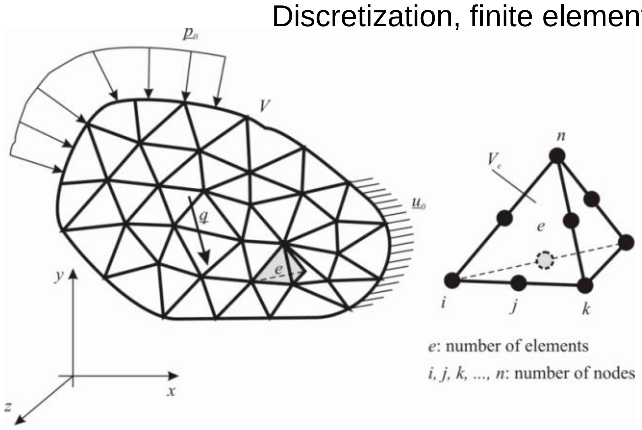
Approximate displacement field

$$\underline{u}_{ei} = \begin{bmatrix} u_{ei} \\ v_{ei} \\ w_{ei} \end{bmatrix}$$

$$\underline{u}_{e} = \begin{bmatrix} u_{ei} \\ v_{ei} \\ w_{ei} \\ \vdots \\ u_{en} \\ v_{en} \\ w_{en} \end{bmatrix} = \begin{bmatrix} \underline{u}_{ei} \\ \underline{u}_{ej} \\ \vdots \\ \underline{u}_{en} \end{bmatrix}$$

Discretization, finite element



$$\underline{u}_{ei}(\underline{r}) = \underline{\underline{N}}_{ei}(\underline{r}) \cdot \underline{u}_{ei} = \begin{bmatrix} N_{eixx}(\underline{r}) & N_{eixy}(\underline{r}) & N_{eixz}(\underline{r}) \\ N_{eiyx}(\underline{r}) & N_{eiyy}(\underline{r}) & N_{eiyz}(\underline{r}) \\ N_{eizx}(\underline{r}) & N_{eizy}(\underline{r}) & N_{eizz}(\underline{r}) \end{bmatrix} \cdot \begin{bmatrix} u_{ei} \\ v_{ei} \\ w_{ei} \end{bmatrix}$$

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CAD Book

12. Application of Finite Element Method

Author: Dr. István Oldal
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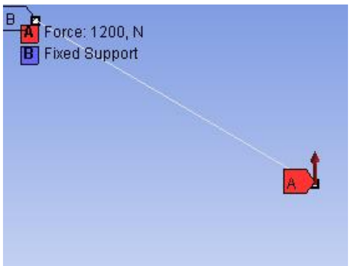
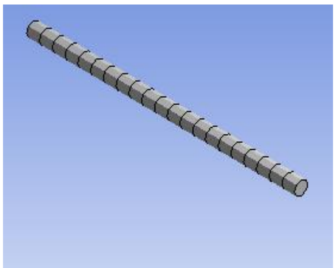
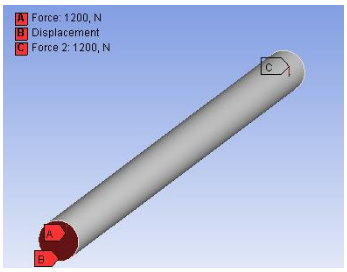
Beam with circular cross section

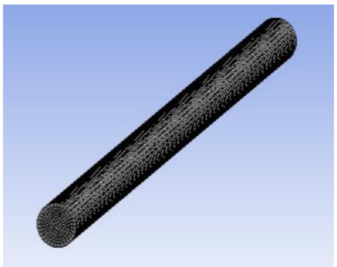


1D model and its mesh





3D model and its mesh



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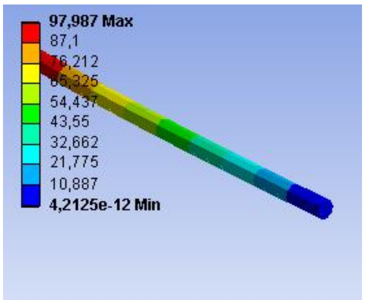
Result comparison

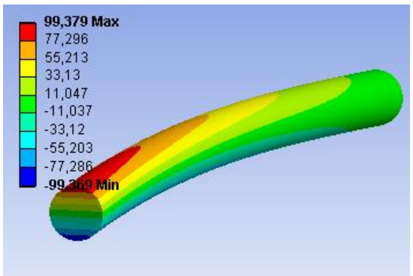
1D model



Exact solution:

$$\sigma = \frac{M_h}{K} = \frac{32 \cdot F \cdot l}{d^3 \pi} = \frac{32 \cdot 1200 N \cdot 1000 mm}{50^3 mm^3 \cdot \pi} = 97{,}78 MPa$$

3D model



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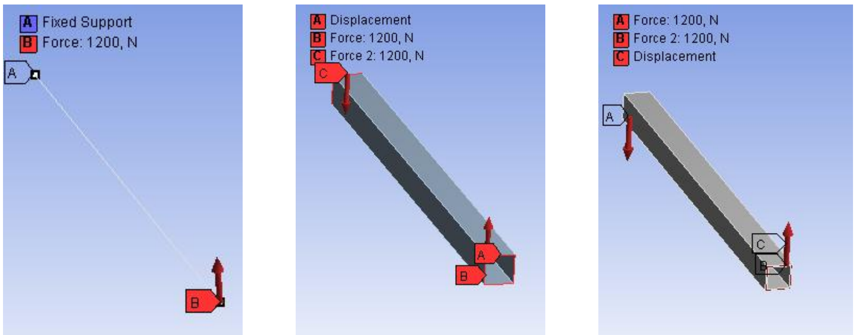
Modeling of thin-valled beams



Geometric model of beam with its constraints and loads

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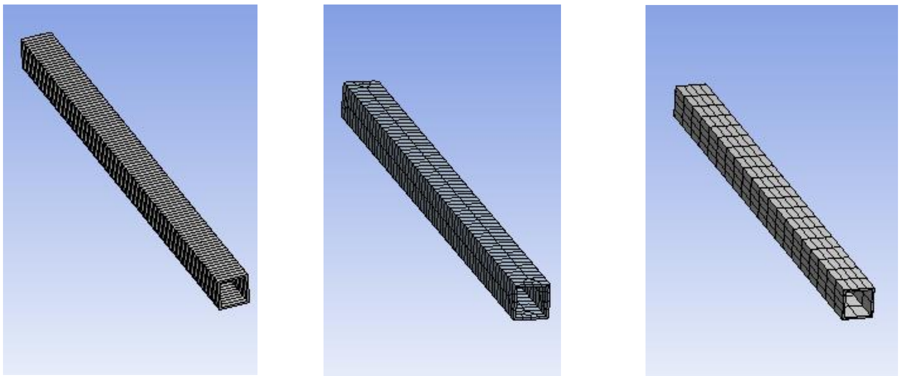
Modeling of thin-valled beams



1D, 2D and 3D finite element models

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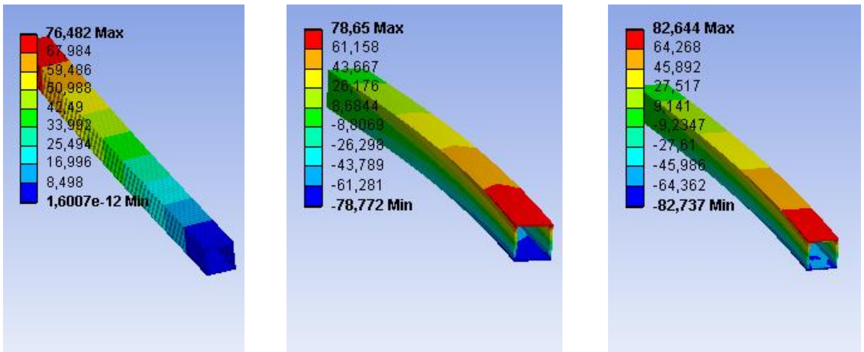
Modeling of thin-valled beams

Exact solution: $$\sigma = \frac{M_h}{I_z} e = \frac{6 \cdot F \cdot l \cdot a}{a^4 - (a-2v)^4} = \frac{6 \cdot 1200N \cdot 1000mm \cdot 60mm}{60^4 mm^4 - 52^4 mm^4} = 76{,}48 MPa$$



Calculated stresses of 1D, 2D and 3D models in MPa

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Modeling of thin-walled open cross section beams

Examined U section: 100x100x4 mm
Section length: 1000 mm
Load force: 1200 N

Maximal calculated stress:

$$\sigma = \frac{M_h}{I_z} e = \frac{1200N \cdot 1000mm}{2103829mm^4} 50mm = 28{,}52MPa$$

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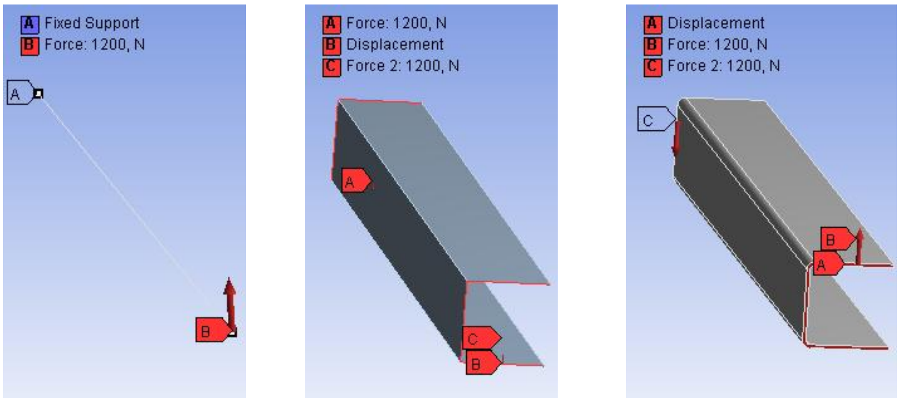
Modeling of thin-walled open cross section beams



Geometric model of beam with its constraints and loads

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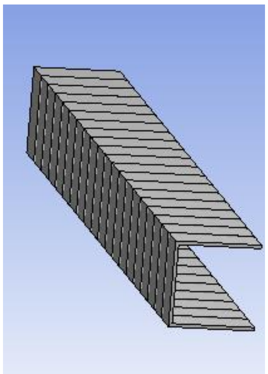
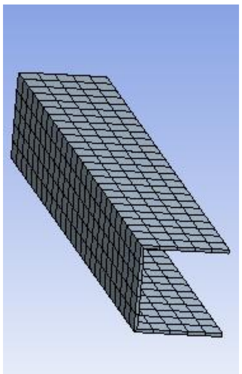
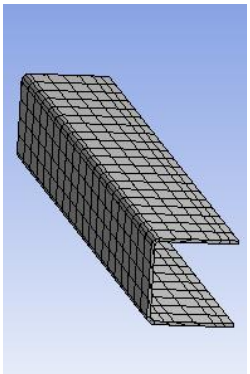
Modeling of thin-walled open cross section beams



20 elements
41 nodes



375 elements
1206 nodes



420 elements
3148 nodes

1D, 2D and 3D finite element models

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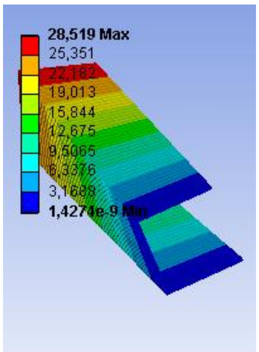
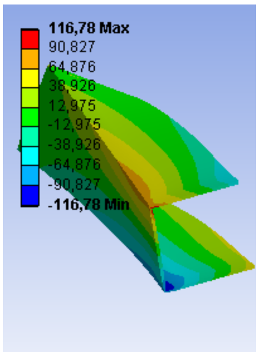
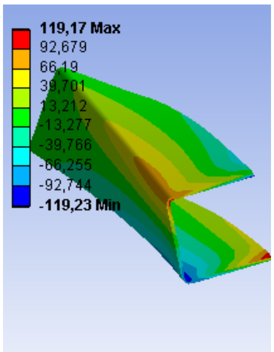
Modeling of thin-walled open cross section beams







20 elements
41 nodes

375 elements
1206 nodes

420 elements
3148 nodes

1D, 2D and 3D finite element models

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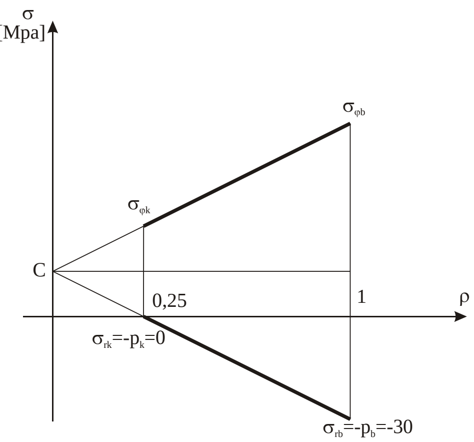
Modeling of thick-walled cylinders, tubes

- Analytical model

$$\rho = \left(\frac{r}{r_b} \right)^2$$

Relative reciprocative radius, where:

r: the radius of the tube (variable)
r_b: the internal radius of the tube



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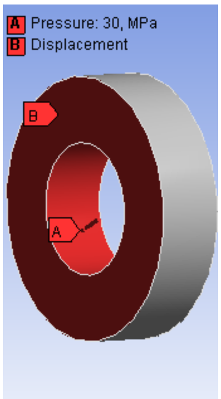
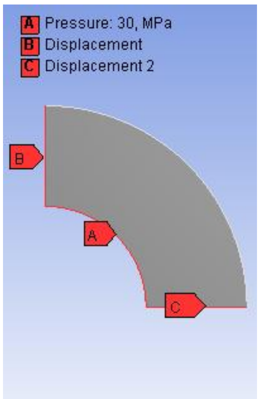
Modeling of thick-walled cylinders, tubes

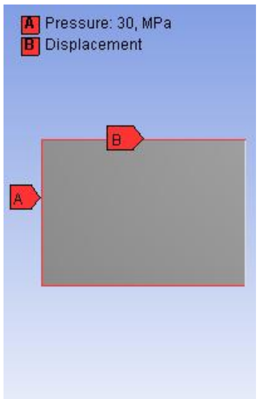
- Finite element models



Short segment



One fourth of cross section



Longitudinal section

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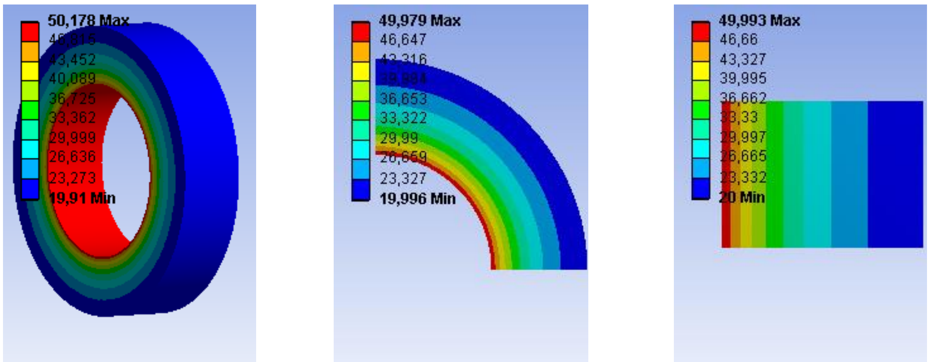
Modeling of thick-walled cylinders, tubes



Tangential stresses calculated by 3D, 2D planar and 2D axis-symmetric models

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CAD BOOK

13. Integration of CAx systems

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- Dölner, Germot – Kelner, Peter: Digital mock-up and rapid prototyping in automotive product development; J. of Integrated Design and Process Science, 2000/3.
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- Kalpakjian, Serope; Schmid, Steven (2006), Manufacturing engineering and technology (5th ed.), Prentice Hall, p. 1192.

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13.1 CAX SYSTEMS

The computer aided design (CAD), the manufacturing (CAM) and the engineering (CAE) are not a stand alone systems, they don not work separately, but they compose one system in order to solve an engineering problem. Lot of engineers work with this adaptable system, often they are in different place. During the corporate work the cooperation realize in different levels.

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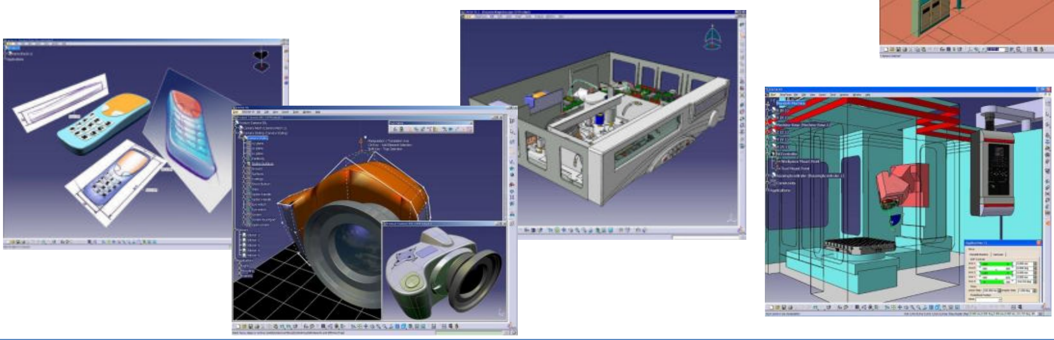
CAx definition

The aim of the development of CAD systems is to increase and extend the capacity and ability of the computer aided design.

It has three way:
- Develop the modelling methods and technology
- Develop special design modules,
- Develop software tools to process the CAD data.



The aim of the development of CAD systems is to increase and extend the capacity and ability of the computer aided design.
It has three way:
- Develop the modelling methods and technology
- Develop special design modules, which able to accelerate the design process in special engineering areas,
- Develop software tools to process the CAD data, which are extend the use of the models.

Computer-aided technologies (CAx) is a broad term that means the use of computer technology to aid in the design, analysis, and manufacture of products.
Advanced CAx tools merge many different aspects of the product lifecycle management (PLM), including design, finite element analysis (FEA), manufacturing, production planning, product testing with virtual lab models and visualization, product documentation, product support, etc. CAx encompasses a broad range of tools, both those commercially available and those proprietary to individual engineering firms.

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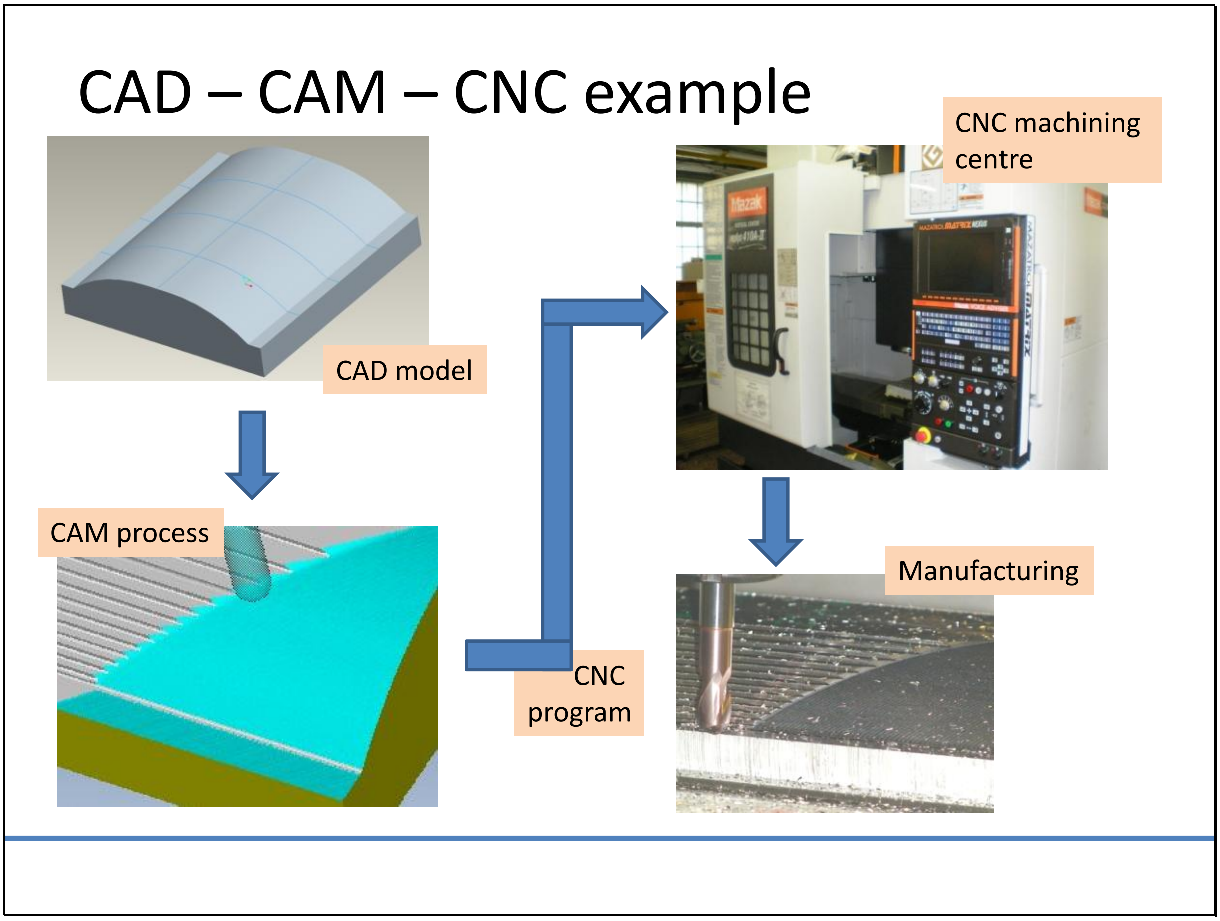

The importance of the development can be seen in case of machining.
The complicated product geometry needs advanced CAD modelling techniques, but the manufacturing required a CNC machine. The CNC program for the manufacturing is created by the CAM system.
The prime mover of the development is the costumer requirement. The complicate part geometry needs CNC machining, the CNC machining needs CNC program, which can be generated by CAM system, but the CAM systems needs the accurate CAD model.

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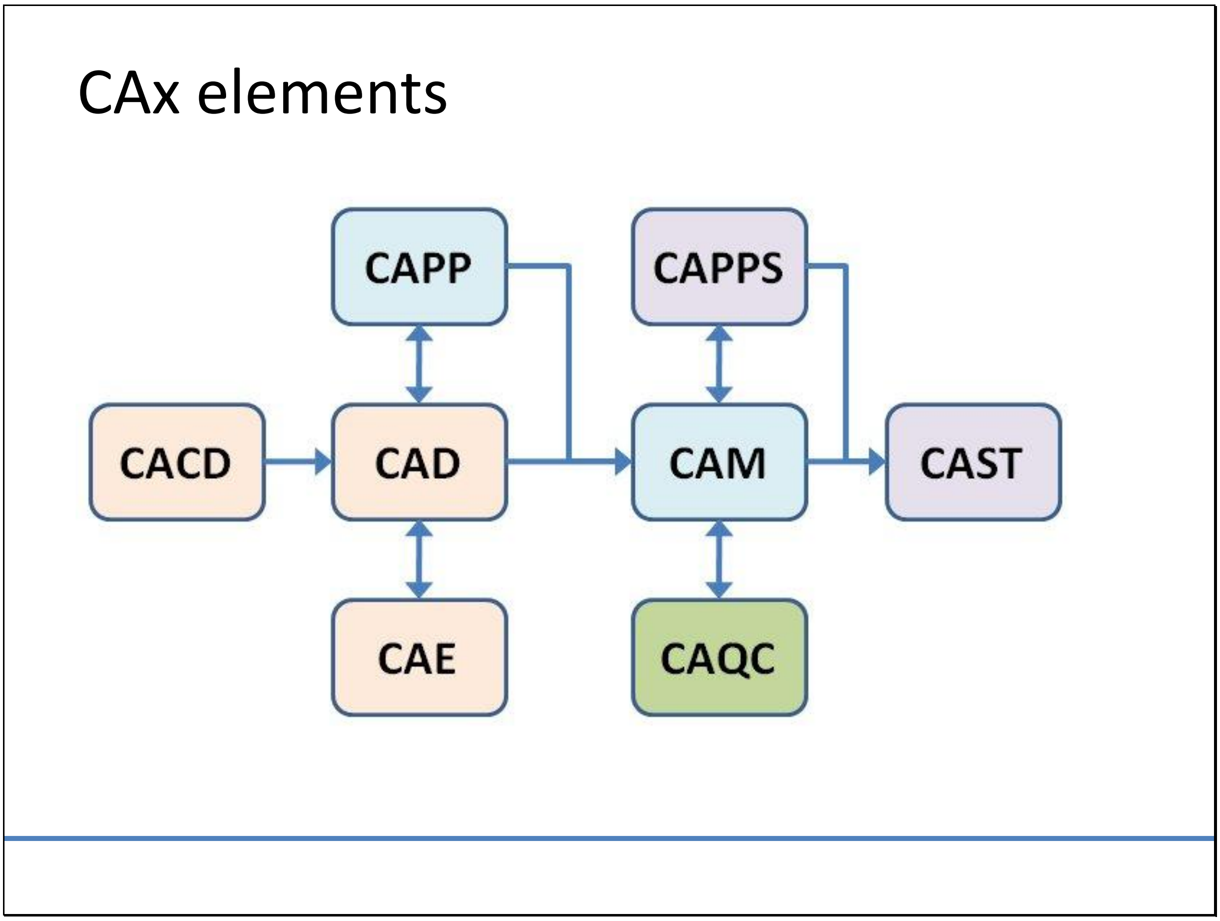

The figure shows the connection of the different CAx technologies.
The first stage is the computer aided conceptual design (CACD), which help to collect and draw up the beginning requirements, design concepts and ideas. The computer aided design (CAD) system ensure a tool for create a detail product models and drawings. The computer aided engineering (CAE) systems support the design process with lot of different simulation and computation solutions. Based on the CAD models the CAM (computer aided manufacturing) systems can design the cutting tool path and NC programs for the production. The connection between the CAD and CAM systems is the computer aided process planning (CAPP), the task of the CAPP is to identify the steps of manufacturing process and define all details of it. The manufacturing process has a close connection to the computer aided quality control (CAQC), which means primary the support of coordinate measuring technique. The manufacturing requirement planning, the scheduling of production process are the essential elements of the production management, which are supported by computer aided production planning and scheduling (CAPPS). The logistic tasks can be designed by the help of computer aided storage and transportation (CAST) system.

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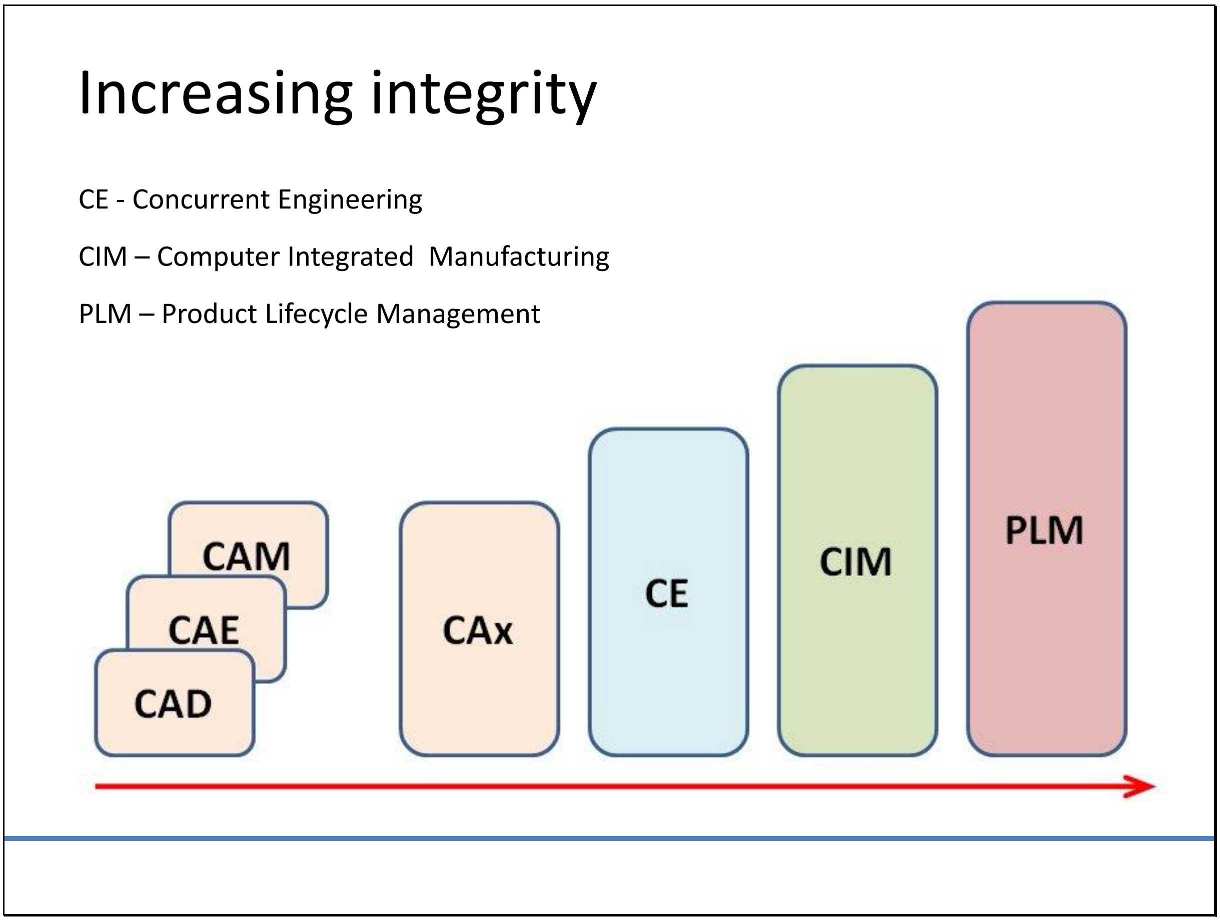

The integration of CAx system has different levels and these levels has an evolution.

Concurrent engineering (CE) is a work methodology based on the parallelization of tasks. It refers to an approach used in product development in which functions of design engineering, manufacturing engineering and other functions are integrated to reduce the elapsed time required to bring a new product to the market.

Computer-integrated manufacturing (CIM) is the manufacturing approach of using computers to control the entire production process. This integration allows individual processes to exchange information with each other and initiate actions. Through the integration of computers, manufacturing can be faster and less error-prone, although the main advantage is the ability to create automated manufacturing processes. In a CIM system functional areas such as design, analysis, planning, purchasing, cost accounting, inventory control, and distribution are linked through the computer with factory floor functions such as materials handling and management, providing direct control and monitoring of all the operations.

Product lifecycle management (PLM) is the process of managing the entire lifecycle of a product from its conception, through design and manufacture, to service and disposal. PLM

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integrates people, data, processes and business systems and provides a product information backbone for companies and their extended enterprise. PLM describes the engineering aspect of a product, from managing descriptions and properties of a product through its development and useful life. The core of PLM (product lifecycle management) is in the creations and central management of all product data and the technology used to access this information and knowledge.

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13.2 INTEGRATION OF CAX SYSTEMS

The CAx integration has two way:

- Some CAD function build up to other applications,
- CAD systems integrate other systems.

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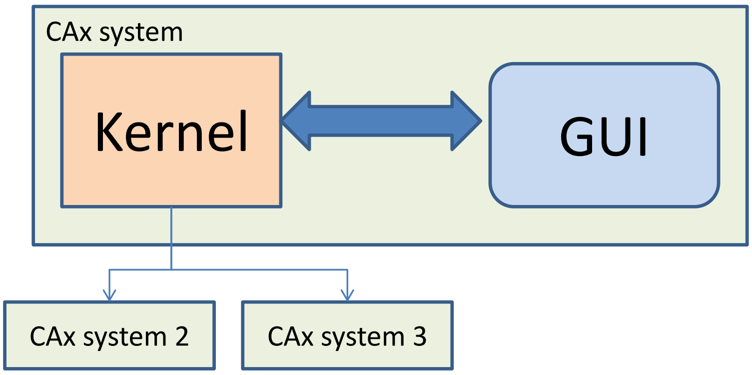
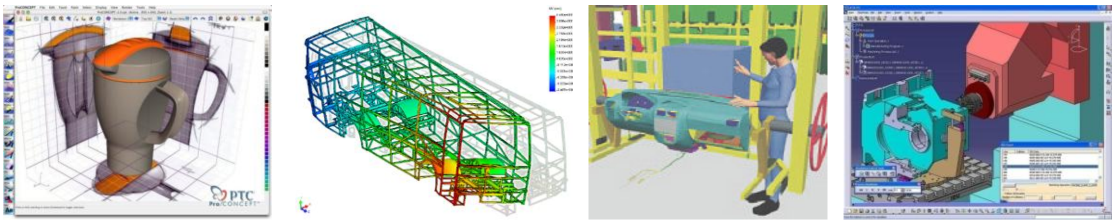
Graphic kernels





During the evolution of the CAD systems the development of the graphic kernel, which perform all calculation tasks, and the development of user interface are divided. This idea ensures the independent development of the mathematical methods and the GUI design.

The tasks of the graphic kernel are:

- Display and manage the 3D objects,
- Communication inside the system,
- Communication with other applications.

The development of a graphic CAD kernel is a very complicated and expensive task. Lot of CAx applications need a limited service of a CAD system, but the integration of a complete CAD system is impossible and mindless, so graphic kernels became commercial products.

The typical application of the graphic kernels are :

- CNC simulation,
- Design application,
- Simulation (visualisation of the results),
- Preview module of PLM system,
- CAD model converter software.

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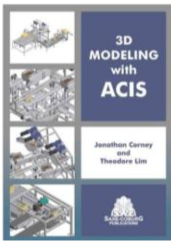
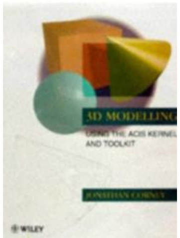
3D graphic kernels

- ***ACIS***:
  - Spiral Corporation (Dassault Systemes, CATIA)
  - Wire frame, surface and solid modelling
  - Functions: 3D modelling, managing and display
  - 1989. v1. ⇨ 2010. v21.
  - OP sys: Windows, Apple OS, SunSolaris, Linux
  - Extension: **.SAT**





The ACIS kernel is the property of Spiral Corporation, which is the part of the Dassault Systemes. The kernel is developed in object oriented C++ environment. The ACIS support the hybrid application of the wireframe, the surface and the solid models.

The first version was published in 1989, and in 2010 the 21st version was published. The kernel is developed for Windows, Apple OS, SunSolaris and Linux operation systems. It has three part: 3D modelling, managing the models and display the 3D models.
The extension of the kernel is ***.SAT***.

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3D graphic kernels

- ***Parasolid***:
  - ShapeData (Siemens PLM (former UGS), NX)
  - 3D surface and solid
  - 2D object set is weak
  - Extension: **.X_T**

The Parasolid kernel is the property of Siemens PLM (former UG).
It support the hybrid surface and solid modelling.
The ability for 2D representation is weak.
The extension of the kernel is ***.X_T***.

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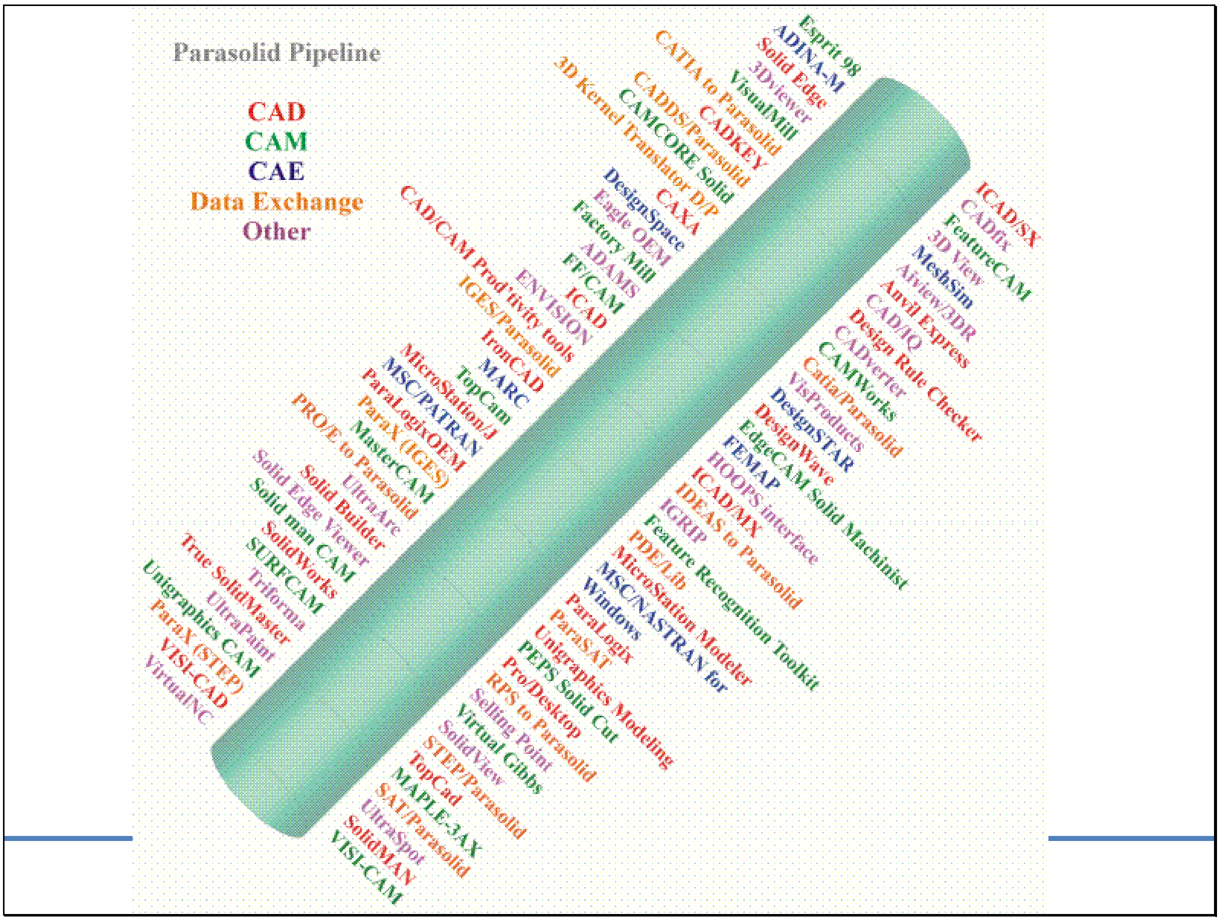

This picture shows the list of CAx application, which use Parasolid kernel. The list contains CAD, CAM, CAE, data exchange and other applications.

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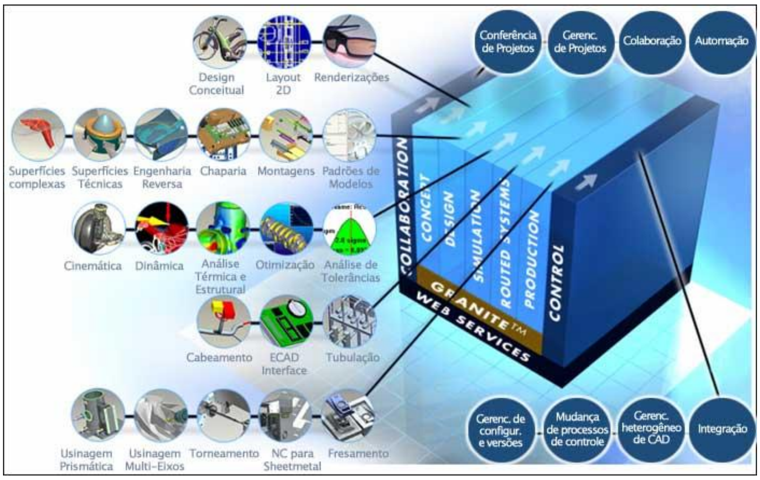
3D graphic kernels

Built on PTC® GRANITE

- ***Granite***:
  - PTC (Pro/Engineer)
  - 3D surface and solid
  - Op. Sys: Windows, SunSolaris, Linux
  - Extension: **.G**



The Granite kernel is the property of PTC. The kernel is developed for Windows, SunSolaris and Linux operation systems.
It support the hybrid surface and solid modelling.
The extension of the kernel is ***.G***.

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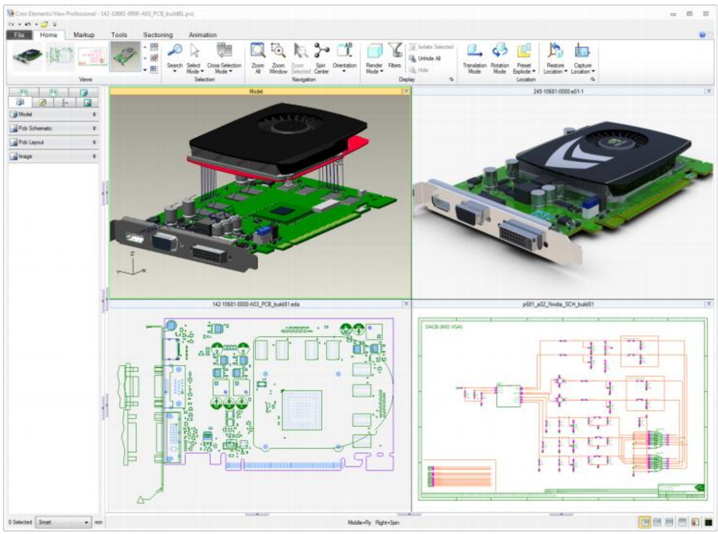
Origin of the special moduls

- Inner development based on users' requirement
- Fully integration of an independent software tool



There is 3 levels of the integration of CAx application:
The first, when independent software solutions work separately. This is a very low level cooperation, it was typical in 70's and 80's. The communication between the software was performed by different file formats. The results were displayed in the CAD system.

The next step was the integrated independent software solution, were the communication is automatic, but the executed software has different user interface, and in general this software has an independent developer.

The third way, when the user can use a fully integrated special module. In that case there is a same user interface, and same distributor. The communication is automatic, the user just starts the special application in the menu system.
The special modules can be born in two ways.
If a large user (like automotive of airplane company) has a special request, the CAD company develops a special module. If it is a successful and usefull, and there is a demand from other users it will become a commercial module.
The other way, when the user demand exists, but the developer found an existing software solution. In that case the bigger company licences the software and integrates or buys the smaller company, and they unite.

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Special CAx moduls

Conceptional design	Casting / moulding	Shoe design
Shape design	Mould design	Printed circuit design
Sheet metal part design	Die design	Ergonomic analisys
Cable design	Sheet metal processing	Airplane design
Pipeline design	Welding process	Automotive design
Kinematic simulation	Coordinate measure machine	
Dinamic simulation		
Static simulation		

Let us see some typical special CAx modules.
The tables contains some interesting application, like the shoe design, but the sheet metal part design module or the mould design module are a very wide range used modules.

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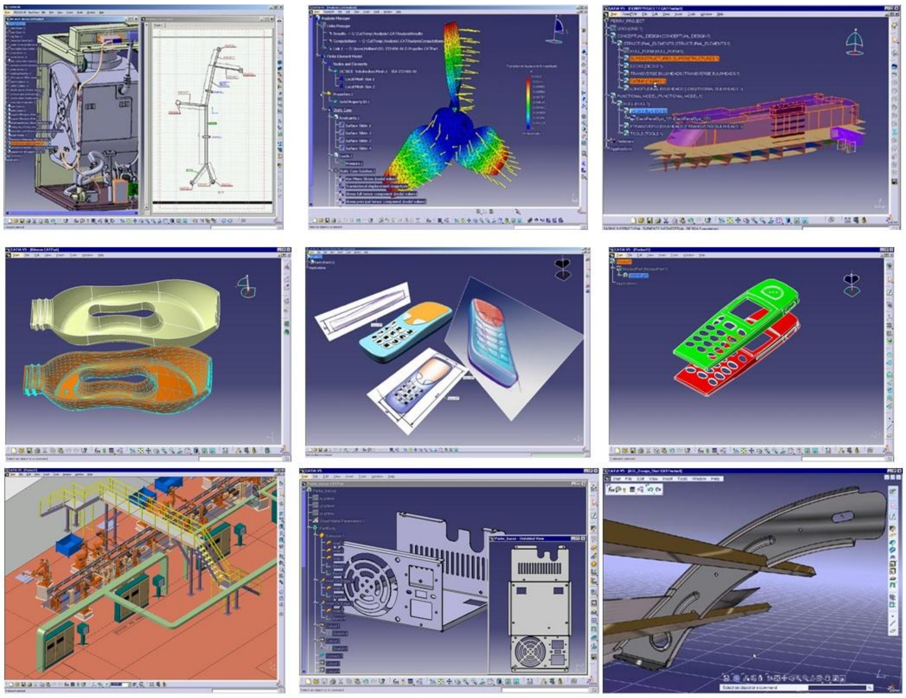
Special CAx modules



The slide shows some pictures about the listed modules from CATIA v5 integrated CAD system.

The advantages of special modules are the next:

- Associative model
- Familiar user interface
- Easy communication and data exchange
- Common product support

The disadvantages of special modules are the next:

- Lower functionalities

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13.3 DATA EXCHANGE

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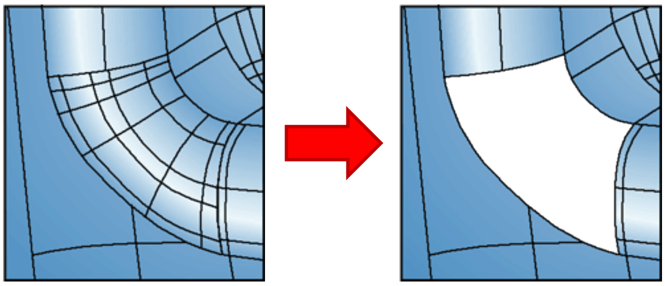
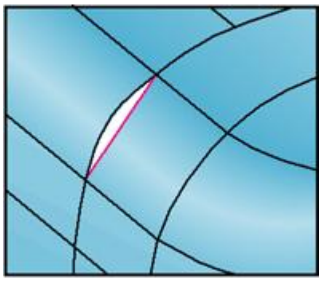
Data exchange between CAx systems

- In native format: CAD system specific format
- In neutral format: standard formats





The data exchange is essential in the integration process and the common work of different systems.
The data exchange can be performed in native file format or in neutral file format.
The native format is a CAD system specific format. We can use it if the systems has a same kernel. In that case there is no file conversation and the CAD model stay parametric and associative.
The neutral formats are standard file formats, which describe the geometric and other data. In that case the data corruption, data loss are frequent problem. The typical faults are the losing of the parametric properties, and the model can not be modified. We lose the model history, and we don't know, how was the model build. Of course sometimes these two deficiencies are good in the viewpoint of data security. The most problematic faults are the geometric problems, like missing surfaces, edges, break of continuity.

The most often used formats are the next:

- DXF
- IGES (.igs)
- VDA/FS (.vda)
- STEP (.stp)
- STL

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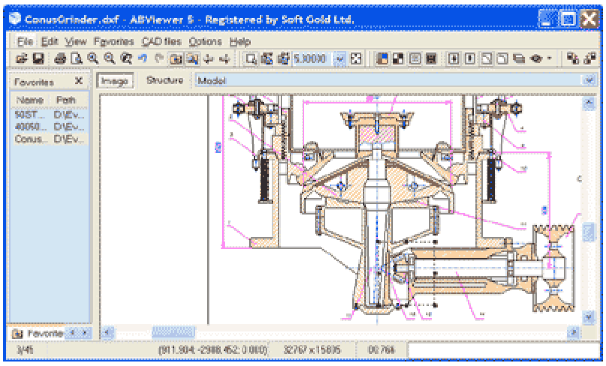
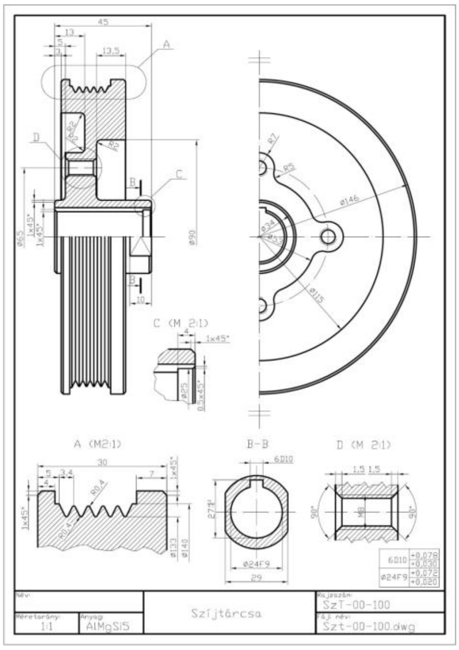
DXF - *Drawing Exchange Format*





AutoCAD DXF (Drawing Interchange Format, or Drawing Exchange Format) is a CAD data file format developed by Autodesk for enabling data interoperability between AutoCAD and other programs. DXF was originally introduced in December 1982 as part of AutoCAD 1.0, and was intended to provide an exact representation of the data in the AutoCAD native file format, DWG (Drawing), for which Autodesk for many years did not publish specifications.

Versions of AutoCAD from Release 10 (October 1988) and up support both ASCII and binary forms of DXF. Earlier versions support only ASCII. ASCII versions of DXF can be read with a text-editor. The format was developed for engineering drawings, so it contains the 2D description of geometric elements, like points, lines, circles, texts etc. The DXF is a widely used format in engineering practice to convert engineering drawings or 2D curves into common format.

As AutoCAD has become more powerful, supporting more complex object types, DXF has become less useful. Certain object types, including ACIS solids and regions, are not documented. Other object types, including AutoCAD 2006's dynamic blocks, and all of the objects specific to the vertical-market versions of AutoCAD, are partially documented, but not well enough to allow other developers to support them. For these reasons many CAD

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applications use the DWG format which can be licensed from AutoDesk or non-natively from the Open Design Alliance.

The DXF is a widely used format in engineering practice to convert engineering drawings or 2D curves into common format.

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IGES (.igs)

- IGES - Initial Graphics Exchange Specification
- 1980. U.S. National Bureau of Standards, (ID: ANSY Y14.26M)
- 1996. last published version (v 5.3)
- Original title: Digital Representation for Communication of Product Definition Data
- Wireframe, surface and solid model
- Engineering drawing and electric circuit drawing

- Objects are identified by numerical code (from 1 to 5000)
- Three type of the objects:
  - Geometrical objects
  - Annotation
  - Structural information

The Initial Graphics Exchange Specification (IGES) defines a neutral data format that allows the digital exchange of information among Computer-aided design (CAD) systems. The official title of IGES is Digital Representation for Communication of Product Definition Data, first published in January, 1980 by the U.S. National Bureau of Standards as NBSIR 80-1978. Many documents referred to it as ASME Y14.26M, the designation of the ANSI committee that approved IGES Version 1.0.

Using IGES, a CAD user can exchange product data models in the form of circuit diagrams, wireframe, freeform surface or solid modeling representations. Applications supported by IGES include traditional engineering drawings, models for analysis, and other manufacturing functions.

The IGES project was started in 1979 by a group of CAD users and vendors, including Boeing, General Electric, Xerox, Computervision and Applicon, with the support of the National Bureau of Standards (now known as NIST) and the U.S. Department of Defense (DoD). The name was carefully chosen to avoid any suggestion of a database standard that would compete with the proprietary databases then used by the different CAD vendors.
Since 1988, the DoD has required that all digital Product Manufacturing Information (PMI) for weapons systems contracts (the engineering drawings, circuit diagrams, etc.) be delivered

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in electronic form, specifically in IGES format. As a consequence, any CAx software vendor who wants to market their product to DoD subcontractors and their partners must support the import (reading) and export (writing) of IGES format files.
An ANSI standard since 1980, IGES has generated enough data to fill warehouses full of magnetic tapes and CD-ROMs of digital PMI for the automotive, aerospace, and shipbuilding industries, as well as for weapons systems from Trident missile guidance systems to entire aircraft carriers. These part models may have to be used years after the vendor of the original design system has gone out of business. IGES files provide a way to access this data decades from now. Today, plugin viewers for Web browsers allow IGES files created 20 years ago to be viewed from anywhere in the world.

After the initial release of STEP (ISO 10303) in 1994, interest in further development of IGES declined, and Version 5.3 (1996) was the last published standard. A decade later, STEP has yet to fulfill its promise of replacing IGES, which remains the most widely used standard for CAx and PMI interoperability.

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IGES geometry objects (entities)

Curves
- 100 circular arc
- 102 composite curve
- 110 line
- 112 parametric spline
- 126 rational B-spline curve

Surfaces
- 108 plane
- 114 spline
- 118 ruled surface
- 120 revolve surface
- 128 rational B-spline surface

Solids
- 150 block
- 154 cylinder
- 156 cone
- 158 sphere
- 160 torus
- 161 revolve solid
- 502 vertex
- 504 edge
- 508 loop
- 510 face
- 514 shell

Other
- 124 transformation matrix
- 132 connect point
- 136 finite element entity

An IGES file is composed of 80-character ASCII records, a record length derived from the punched card era. Text strings are represented in "Hollerith" format, the number of characters in the string, followed by the letter "H", followed by the text string, e.g., "4HSLOT" (this is the text string format used in early versions of the Fortran language). Early IGES translators had problems with IBM mainframe computers because the mainframes used EBCDIC encoding for text, and some EBCDIC-ASCII translators would either substitute the wrong character, or improperly set the Parity bit, causing a misread.

The file is divided into 5 Sections, indicated by a character (S, G, D, P, or T) in column 73. The characteristics and geometric information for an entity is split between two sections; one in a two record, fixed-length format (the Directory Entry, or DE Section), the other in a multiple record, comma delimited format (the Parameter Data, or PD Section), as can be seen in a more human-readable representation of the file.

The pictures show some typical geometric and other entities.

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IGES objects (entities)

Annotation entities

- 206 diameter dimension
- 212 general note
- 216 linear dimension
- 220 point dimension
- 222 radius dimension
- 230 sectioned area

Structure entities

- 302 associativity definition
- 310 text font definition
- 314 colour definition
- 404 drawing entity
- 410 view entity
- 412 rectangular array subfigure
- 414 circular array subfigure

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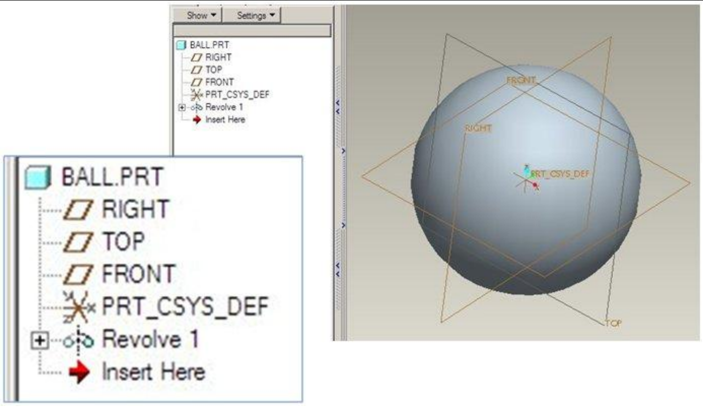
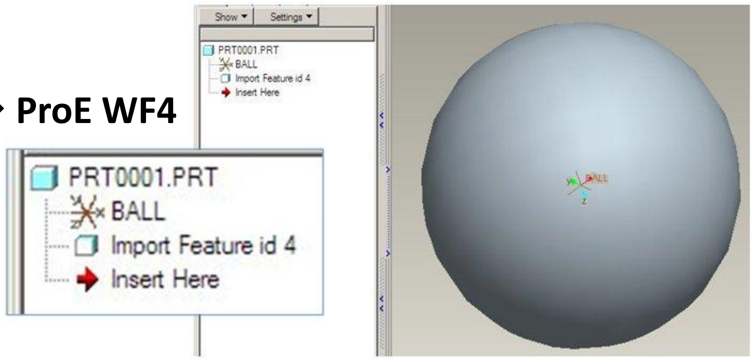
Example

Ø50 sphere
ProE WF4



Ø50 sphere
ProE WF4 ⇨ IGES ⇨ ProE WF4



If we see the 3D model of a ball, we can follow the lists of definition in the history tree. In the first picture there are 3 plane, a coordinate system and a revolved feature. This is parametric, because we can modify the size of the ball, or the contour of the revolved feature.

If we create an IGES export and open it, the model history consists of a coordinate system and an import feature, which is not able to modify, we can't change the size etc. If the original model consists of more feature, the exported model contains only one import feature independently of the complexity of the model.

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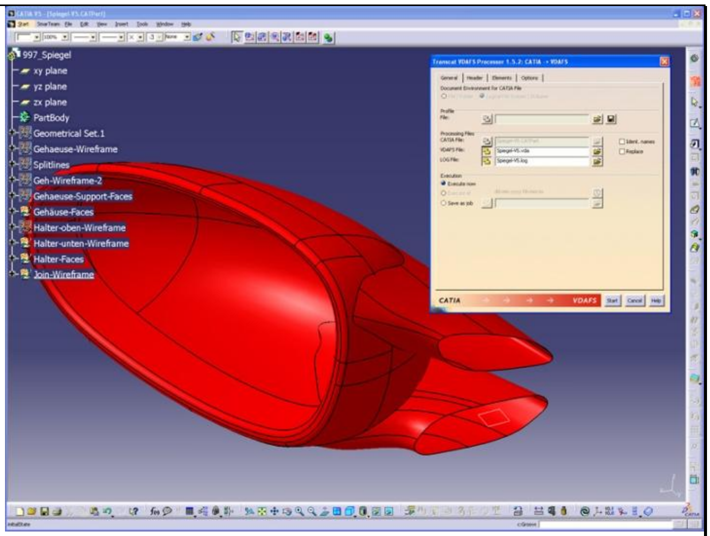
VDA/FS



- "Verband der Automobilindustrie - Flächenschnittstelle„
- „Organisation of the automotive industry - surface translation format"
- German standard (DIN 66301)
- Free form surface representation for automotive industry

VDA-FS is a CAD data exchange format for the transfer of surface models from one CAD system to another. Its name stands for "Verband der Automobilindustrie - Flächenschnittstelle", which translates to the "organisation of the automotive industry - surface translation format". Standard was specified by the German organisation VDA.

The VDA-FS was developed for automotive industry, and it can be describe the free form surfaces. It was developed for correct the initial problems of IGES. The VDA-FS is an ASCII file, which contains the technical data, like name of the model, version number, datum, designer etc. And the geometric data.

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STEP

- ISO 10303 St*andard for the Exchange of Product model data*
- fully description of the product data (not only geometry)
- developer: ISO TC 184/SC4 commitee
- 1st phase started in 1984., first version 1995.
- 2nd phase is finished in 2002. - special industrial application

ISO 10303 is an ISO standard for the computer-interpretable representation and exchange of product manufacturing information. Its official title is: Automation systems and integration — Product data representation and exchange. It is known informally as "STEP", which stands for "Standard for the Exchange of Product model data".

The International standard's objective is to provide a mechanism that is capable of describing product data throughout the life cycle of a product, independent from any particular system. The nature of this description makes it suitable not only for neutral file exchange, but also as a basis for implementing and sharing product databases and archiving.

Typically STEP can be used to exchange data between CAD, Computer-aided manufacturing, Computer-aided engineering, Product Data Management/EDM and other CAx systems. STEP is addressing product data from mechanical and electrical design, Geometric dimensioning and tolerancing, analysis and manufacturing, with additional information specific to various industries such as automotive, aerospace, building construction, ship, oil and gas, process plants and others.

The evolution of STEP can be divided into three release phases. The development of STEP started in 1984 as a successor of IGES, SET and VDA-FS . Today AP 203 Configuration controlled 3D design is still one of the most important parts of STEP and supported by many CAD systems for import and export.

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In the second phase the capabilities of STEP got widely extended, primarily for the design of products in the aerospace, automotive, electrical, electronic, and other industries. This phase ended in the year 2002 with the second major release. A major problem with the APs of the first and second release is that they are too big, have too much overlap with each other and are not sufficiently harmonized.

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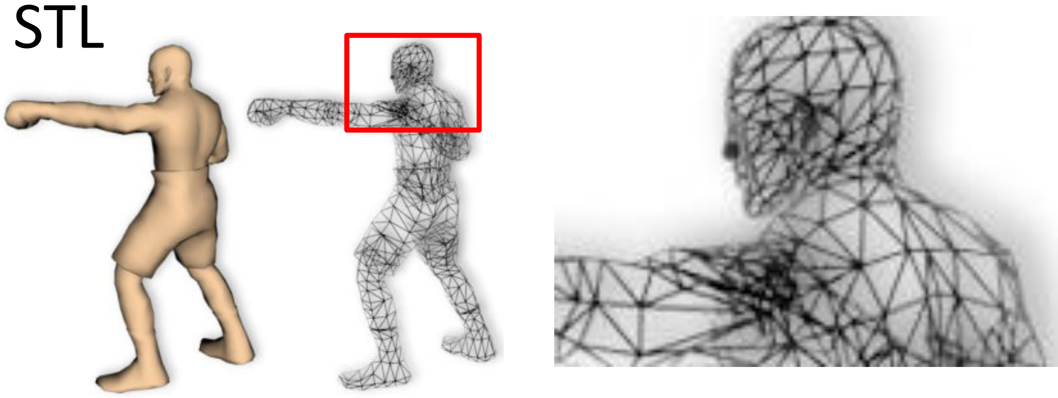
STL



- Developer: 3D System Ltd
- For stereolithographi rapid prototyping
- 3D geometry without colour, texture or other properties
- ASCII or binary format (smaller file size)
- Divide the part surface into small triangles
- Define the coordinate of the corner points and the normal vector

STL is a file format native to the stereolithography CAD software created by 3D Systems. This file format is supported by many other software packages; it is widely used for rapid prototyping and computer-aided manufacturing. STL files describe only the surface geometry of a three dimensional object without any representation of color, texture or other common CAD model attributes. The STL format specifies both ASCII and binary representations. Binary files are more common, since they are more compact.

An STL file describes a raw unstructured triangulated surface by the unit normal and vertices (ordered by the right-hand rule) of the triangles using a three-dimensional Cartesian coordinate system.

In both ASCII and binary versions of STL, the facet normal should be a unit vector pointing outwards from the solid object. In most software this may be set to (0,0,0) and the software will automatically calculate a normal based on the order of the triangle vertices using the 'right hand rule'. Some STL loaders (eg the STL plugin for Art of Illusion) check that the normal in the file agrees with the normal they calculate using the right hand rule and warn you when it does not. Other software may ignore the facet normal entirely and use only the right hand rule. So in order to be entirely portable one should provide both the facet normal and order the vertices appropriately - even though it is seemingly redundant to do so. Some other software use the normal for shading effects, so the "normals" listed in the file are not the true facets' normals.

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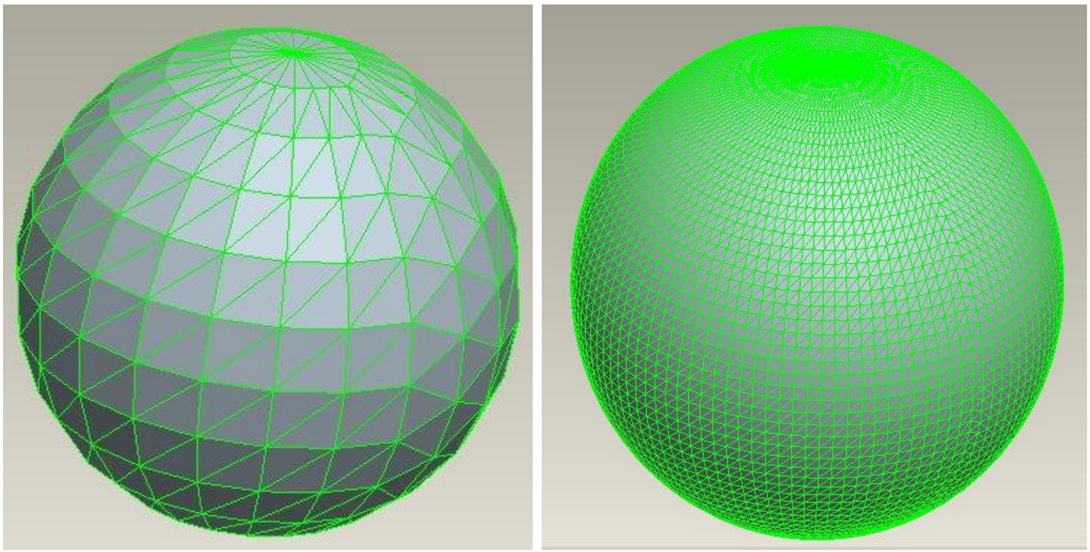

If the export the previous sphere to STL format and reopen it, the triangles can be seen. The dimension of the triangles can be adjusted, and we can improve the surface quality of the model.

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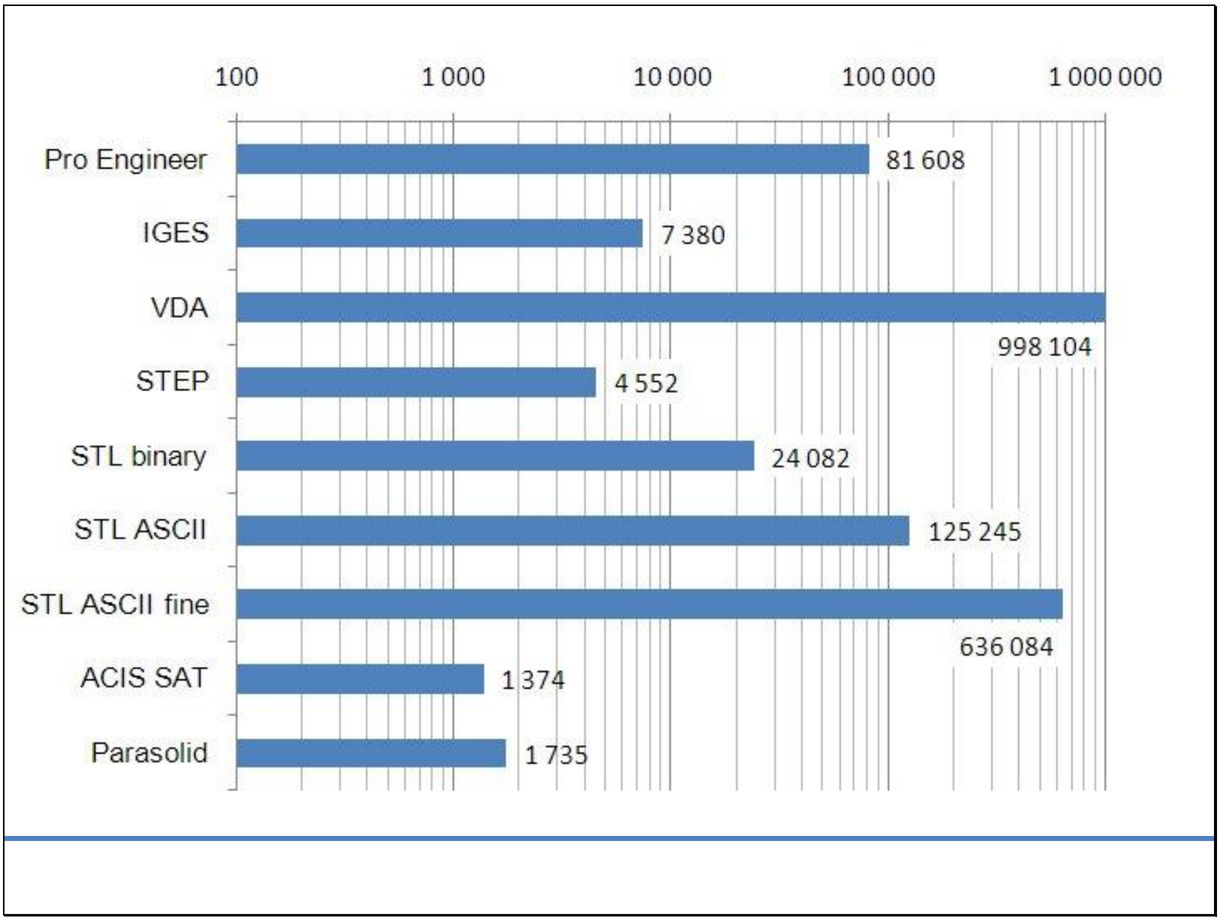

The diagram shows the size of the model of the sphere in case of different file formats in logarithmic scale.
The original model was created in ProEngineer WF4. The size of the file is about 82 kB. The VDA representation is about 1MB, this is the largest version. The smallest file was the kernel format (ASIC and Parasolid). The STL files' size depends on the density of the corner points of the triangles.
The IGES and STEP files have more then 10 times smaller size, than the original model.

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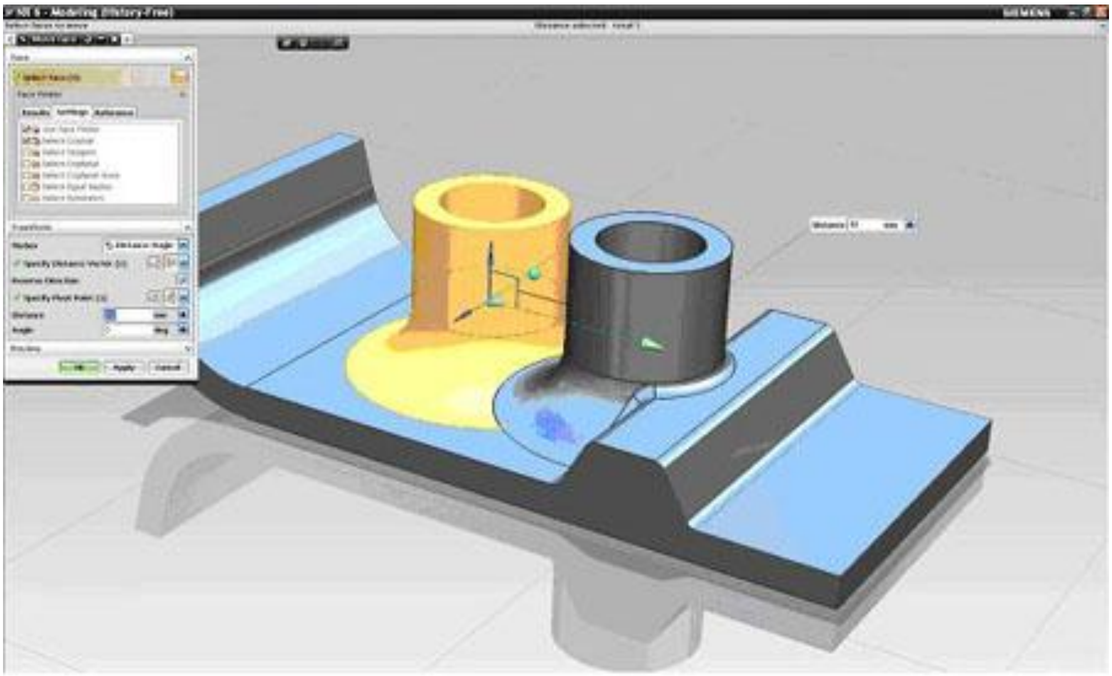
Synchronous modelling



The synchronous modelling is the patented technology of Siemens PLM (NX, Solid Edge) since 2008.
It works on Parasolid kernel, and it ensure the parametric modification of a non parametric model and the direct modelling without sketches, and there is no model history. This technology correct the defects of manipulation of neutral models.

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13.4 CAD LIBRARIES

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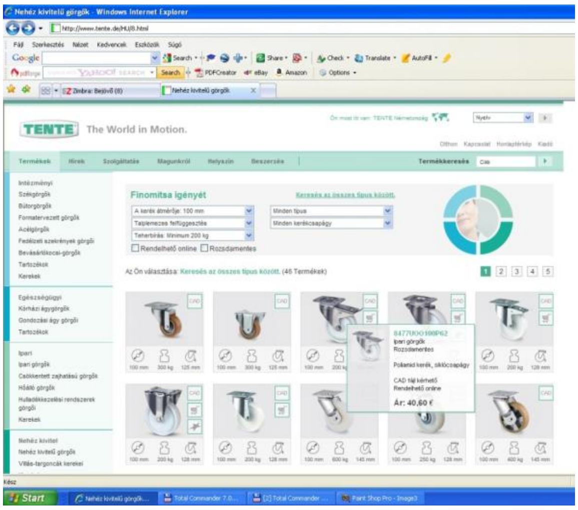
CAD libraries

- Standard parts, integrated user interface

- Supplier specific off-line catalogue
  - Integrated
  - Not-integrated
- Supplier specific on-line catalogue
  - Sometimes registration required



During the engineering design shall be sought to reduce the own designed parts. The aim of this rule is to increase the productivity of the design process, and rationalize the manufacturing and assembly. The reused components can be standard parts, like screws or commercial components, like bearings, motors, hydraulic or pneumatic components etc.

The productivity of the design process can be further increased by use of electronic catalogues and the product will be build up from downloaded components.

This catalogues or libraries can be an integrated component of a CAD system, with the common user interface, and in general they contain standard elements.

If we need a special part or component, we have to use the supplier specific catalogues, because we will order this part from the selected company and we need the accurate model of the element.

The catalogue can be an off-line catalogue, which have to be installed to the computer, and it can be an integrated with the CAD system, or stand alone.

The other possibilities is the on-line catalogue, which can be use via internet and don't need installation. In that case the database is up-to-date, but without internet it can't be used, and sometimes the use of it needs registration.

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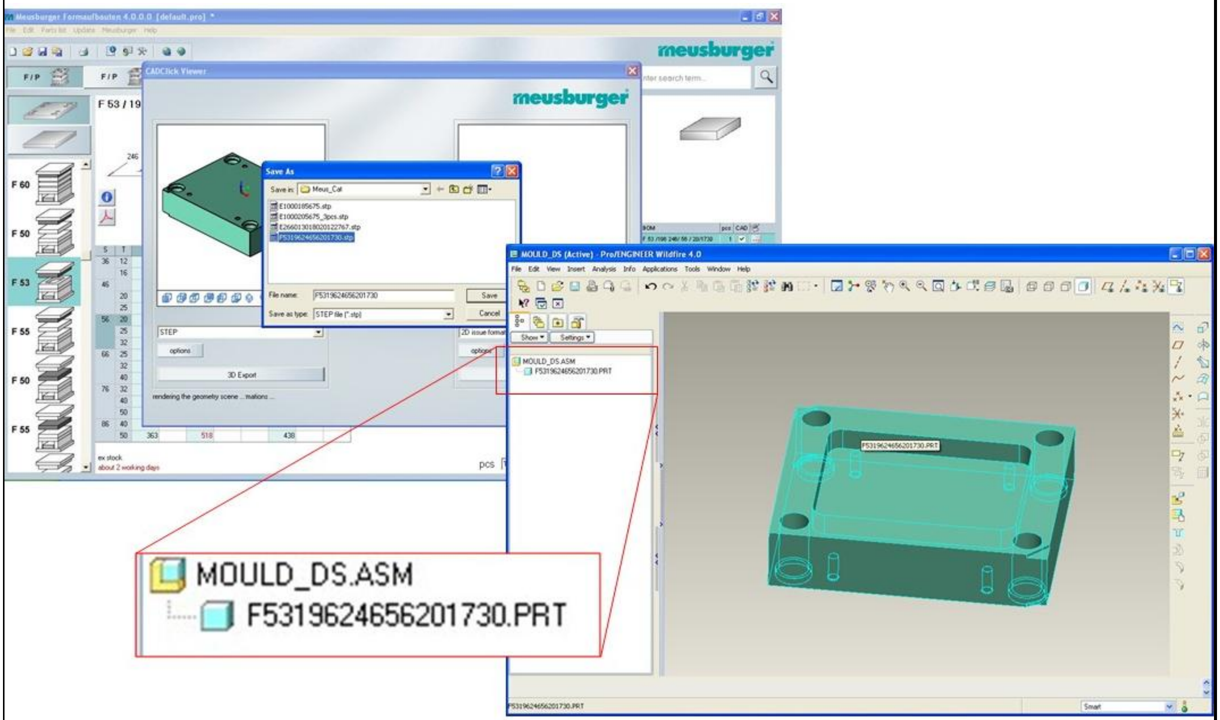

The picture shows an example from the Meusburger mould component catalogue.
After the selection of the required plate, the 3D model can be downloaded and open in a CAD system. As the model tree shows, the structure and the features of the model cannot see, the 3D model consists of only one feature. The model is not parametric.

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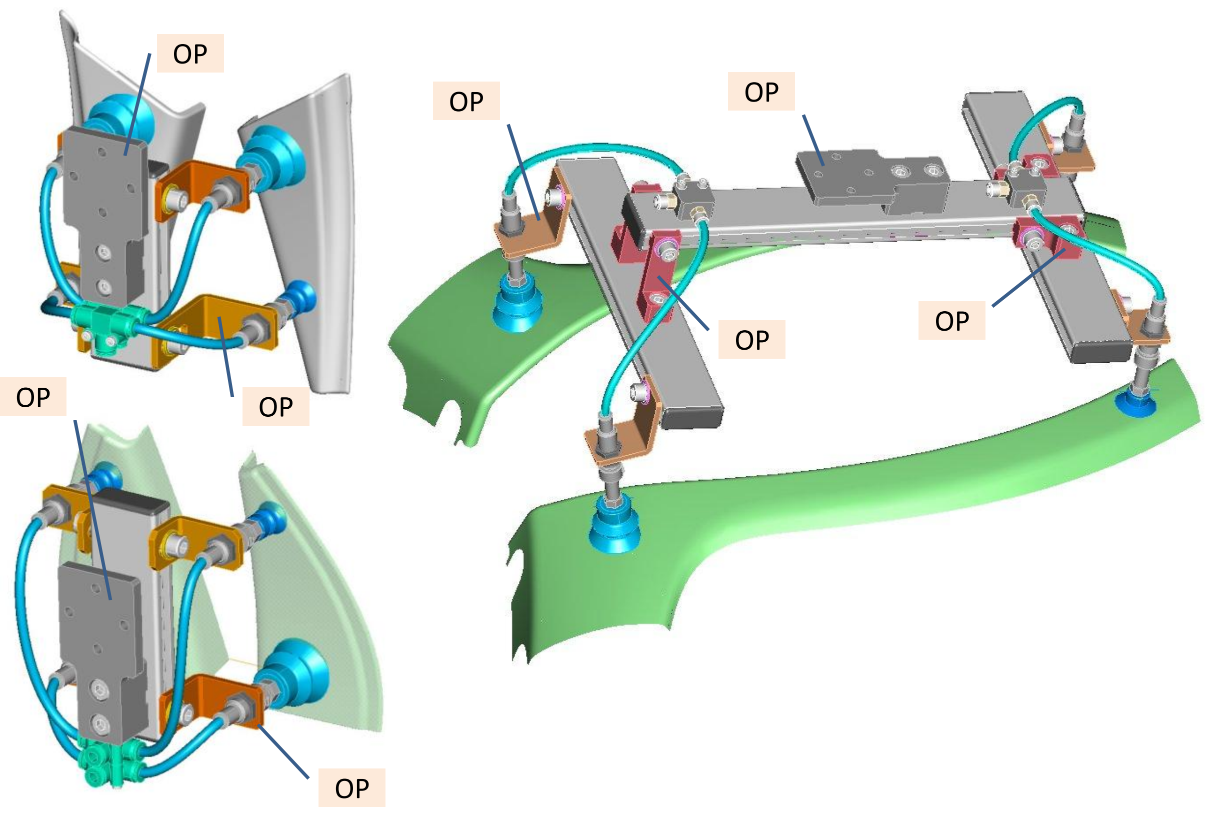

These robot grippers were designed for handling of plastic parts.
The number of original parts (OP) is very low, because lot of elements can be find in suppliers' digital catalogues, and the downloaded 3D CAD model were assembled to the product.

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13.5 DIGITAL MOCK-UP

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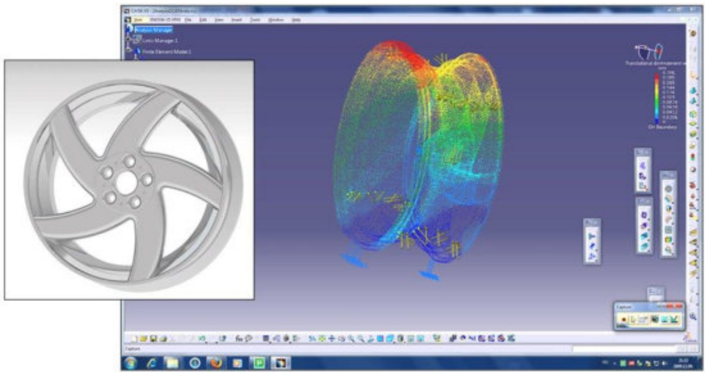
Digital mock-up definition

Digital mock-up (DMU) is a concept that allows the description of a product, usually in 3D, for its entire life cycle.

Digital mock-up is enriched by all the activities that contribute to describing the product.



Digital mock-up (DMU) is a concept that allows the description of a product, usually in 3D, for its entire life cycle. Digital mock-up is enriched by all the activities that contribute to describing the product.

The product design engineers, the manufacturing engineers, and the support engineers work together to create and manage the DMU. One of the objectives is to have an important knowledge of the future or the supported product to replace any physical prototypes with virtual ones, using CAx techniques. As an extension it is also frequently referred to as Digital Prototyping or Virtual Prototyping. These two specific definitions refer to the production of a physical prototype, but they are part of the DMU concept.

DMU allows engineers to design and configure complex products and validate their designs without ever needing to build a physical model.

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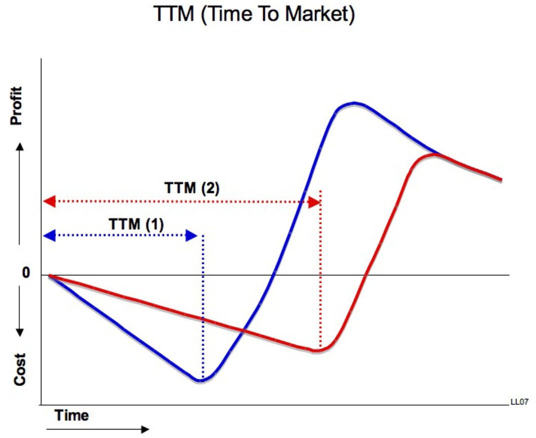
Goal of DMU

- Reduce time-to-market
- Reduce product development costs
- Increase product quality



The main goal and advantages of the using of digital mock-up is the next:

- Reduce time-to-market by identifying potential issues earlier in the design process.
- Reduce product development costs by minimizing the number of *physical prototypes* that need to be built.
- Increase product quality by allowing a greater number of design alternatives to be investigated before a final one is chosen.

Digital mock-up not only reduces the time A&D manufacturers need to develop and validate the product, it also offers more much flexibility in evolving the product design. Moreover, digital mock-up is perfectly suited for the virtual product development environment in which the OEMs and globally dispersed suppliers and partners must collaborate.

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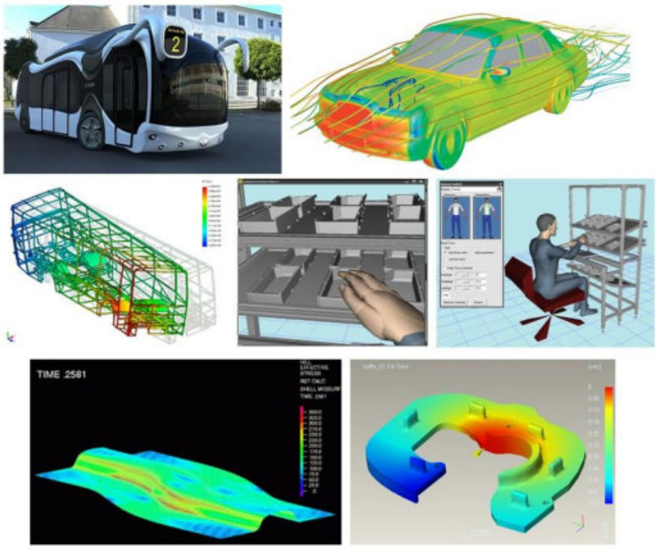
Possible simulation



- Photorealistic visualisation
- Motion analysis
- Collision analysis
- Tolerance analysis
- FEM (Static, Dynamic , Flow,
- Magnetic, Heat)
- Ergonomic analysis
- DfX analysis
- Manufacturing simulation (cutting, casting, moulding, forming)
- Virtual crash test

The digital mock-up ensures to perform several analysis, which were mentioned previously. These analysis are not essential part of the digital mock-up, it only ensures the possibilities for it by all required data.

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CAD Book

14. PDM/PLM systems

Author: Attila Piros

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Introduction

Nowadays the engineering activity not only focuses on the design of the products, creating the related computer models with the necessary manufacturing documentation, but this activity is much broader and it includes more extensive tasks. These tasks have to be resolved in global design groups, i.e. together with engineers in different locations.

The design procedures can fully be covered by computational support. But apart from the obvious advantages, this support also has disadvantages. The exponentially growing electronic data causes many problems in the design procedure.

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Introduction

An engineer has to face the following challenges during the design procedure:

♦ review and handling of an enormous amount of electronic data

♦ tracking the changes of the product data

♦ supporting extended teamwork in time and location

♦ handling of the design related other procedures

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PDM/PLM systems

The PDM (Product Database Management) systems handle only the product data. The PLM (Product Lifecycle Management) systems handle the product and all related data during its lifecycle. A typical PLM system has modular structure.

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Concurrent engineering

Before introducing the design supporting background systems it is useful to give an overview of the engineering environment of these systems, highlighting the area of concurrent engineering.

Concurrent Engineering (or Simultaneous Engineering) is the method of the design, manufacturing, logistic and service procedures. This method arranges all the possible procedures parallel both in time and logical work flow. The advantages of this method are the following:

- ♦ with 30% - 70% shorter development time
- ♦ with 65% - 90% fewer changes in design
- ♦ with 20% - 90% shorter time to market time
- ♦ with 200% - 600% higher product quality
- ♦ with 20% - 110% higher engineering productivity

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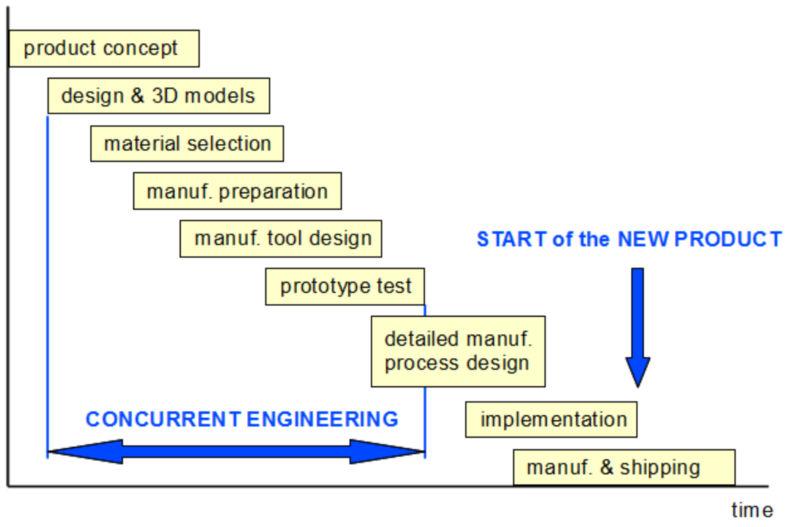
Concurrent engineering

The area of Concurrent Engineering is located between the conceptual design of the product and the design of the manufacturing process.



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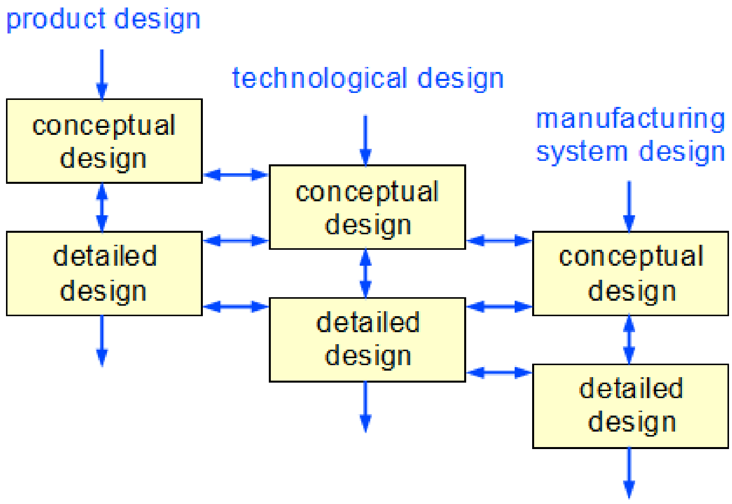
Concurrent engineering

In Concurrent Engineering the human and the infrastructural (IT) resources are used at the same time with parallel execution of the design steps. Here the management of the information flow is very important among the design procedures.



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Product model and information management

The Product Model is the lifecycle model of the product with all the required information for design and manufacturing. Years before this model was a 3-dimensional CAD model, but nowadays it is a highly detailed computer model so called Digital Mock-Up (DMU). The main aim of using a DMU is the replacement of the physical tests with virtual ones. These virtual tests could be following ones:

- ♦ kinematic and dynamic simulations
- ♦ different FEA test (static, fluid dynamic, etc.)
- ♦ ergonomic studies
- ♦ industrial design related studies (surface continuity, reflection tests)
- ♦ fundamental interference test among the built-in components
- ♦ assembly and service simulations

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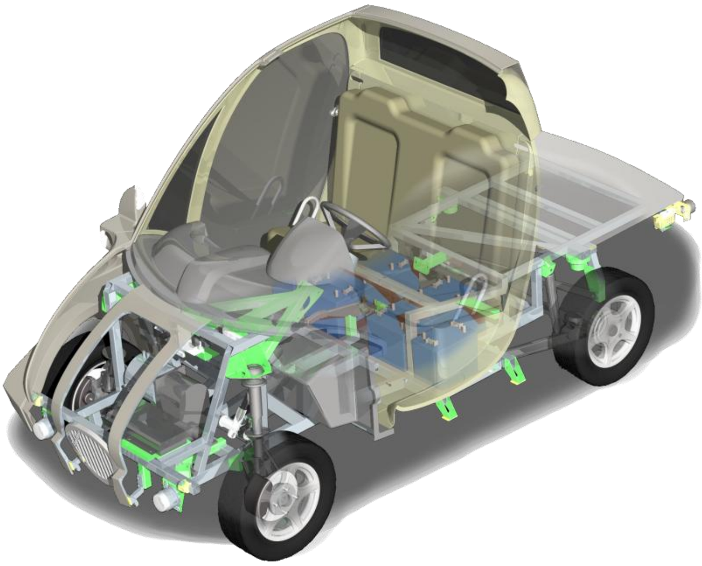
Product model and information management

This kind of Digital Mock-Up can be found in different areas of the industry (defence, aerospace and vehicle industry).



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Product model and information management

The Product Data Management (PDM), with integration of the product data from various sources, helps the engineers and the other participants control the procedure of the product development. Typical product data could be the following:

♦ component identification mark or concerning standard ID

♦ description or vendor ID

♦ applied unit system

♦ cost or purchasing price

♦ material properties

♦ design notes and directives

♦ technological requirements

♦ other related documents (reports, photos, measurement data, etc.)

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Product model and information management

The huge amount of product information can only be stored in databases. These systems provide the structural storage of the product information. The fundamental elements of these systems are the following:

Data

The data is the set of those unstructured facts which can be stored, searched, actualized and restored. These data are typical raw information like measurement data. There are some special types of data:

♦ **software**: this data can manipulate another data

♦ **metadata**: stores the description of another data

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Product model and information management

Information

Information is the set of facts with meaning and evaluated data. The most important function of a database system is extract/retrieve these information from the data.

Database (DB)

Database is the collection of the long turn stored structural information. DB is an integrated data structure storing the objects together with the related information (metadata). The data models of the databases are the following:

♦ **Flat model**: data stored in tables

♦ **Hierarchical model**: data stored in tree structure, good performance in case of parent/child relationship (BOM, catalogues, etc.)

♦ **Network model**: data stored in records, the records organized in groups, cross references among data are enabled

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Product model and information management

Other data models of the databases:

♦ **Relational model**: stores the relationships among data, the data and the relationships stored in different tables, very flexible data model

♦ **Object-oriented model** (Objectional model): the database built from intelligent object supporting the following operations:

- encapsulation: data and the related operations are stored in objects
- inheritance: high level objects (children) inherit properties from low level objects (parents)
- polymorphism: the same command is interpreted in different ways in case of different objects

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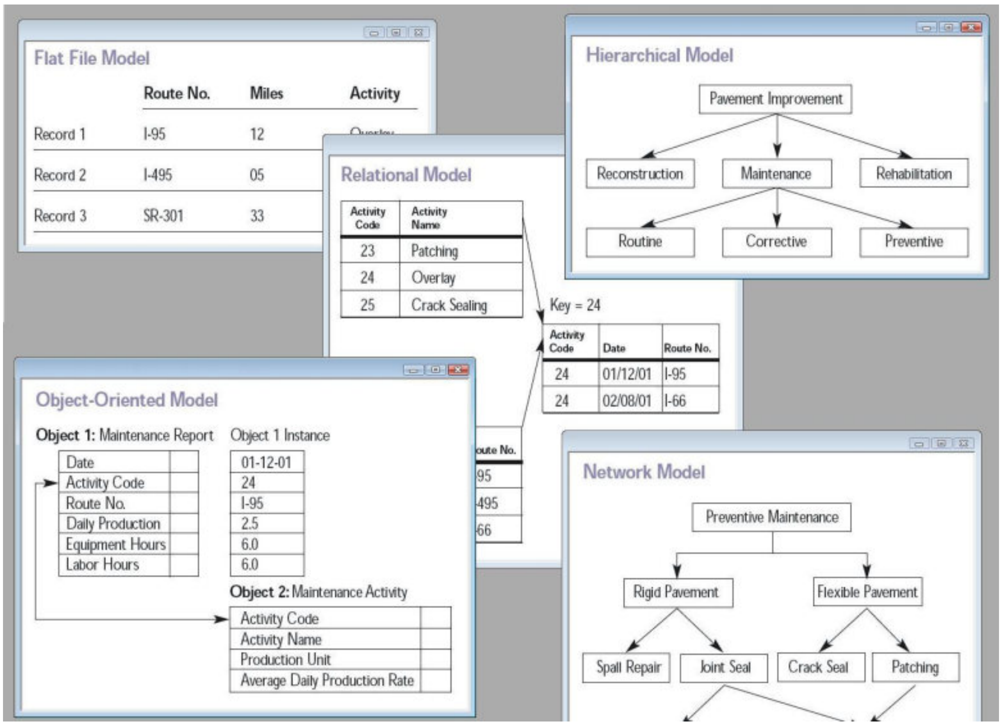
Product model and information management



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Product model and information management

DataBase Management System (DBMS)

DBMS is a software system providing access for the databases and containing various service functions. The fundamental functions of a DBMS are the following:

♦ **indexing**: a method for increasing the speed of data search

♦ **supporting transactions**: controlled manipulating of several data in the same time

♦ **replication**: continuous synchronizing of more database instances, automatic switching in case of the error of the primary database

♦ **security functions**: support of the encryption, user access rights and logging the activities

♦ **locking**: DBMS locks the related data until a transaction is successfully closed

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Product model and information management

The additional functions of a DBMS are the following:

♦ **query ability**: supporting the complex, sometimes nested queries

♦ **backup and replication**: supporting the unified database structures even in case of far geographical locations

♦ **rule enforcement**: decreasing the errors caused by manual input

♦ **advanced security**: controlling the data access, remove functions both on user and user group levels

♦ **change and access logging**: tracking the data manipulation

♦ **automated optimization**: statistically based automated setup procedure to increase the performance of the DBMS

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Product Database Management (PDM)

PDM systems provide the structural storage for all the product related data. The main functions of the PDM systems are the following:

♦ **heterogeneous data handling**: storage of heterogeneous product data (CAD files, Office documents, e-mails, etc.)

♦ **quick information search**: in case of any product data

♦ **visualization**: displaying the up to date CAD data (previews, exploded views, etc.)

♦ **product structure handling**: displaying and exporting the product structures in customizable formats (Bill of Material (BOM) lists)

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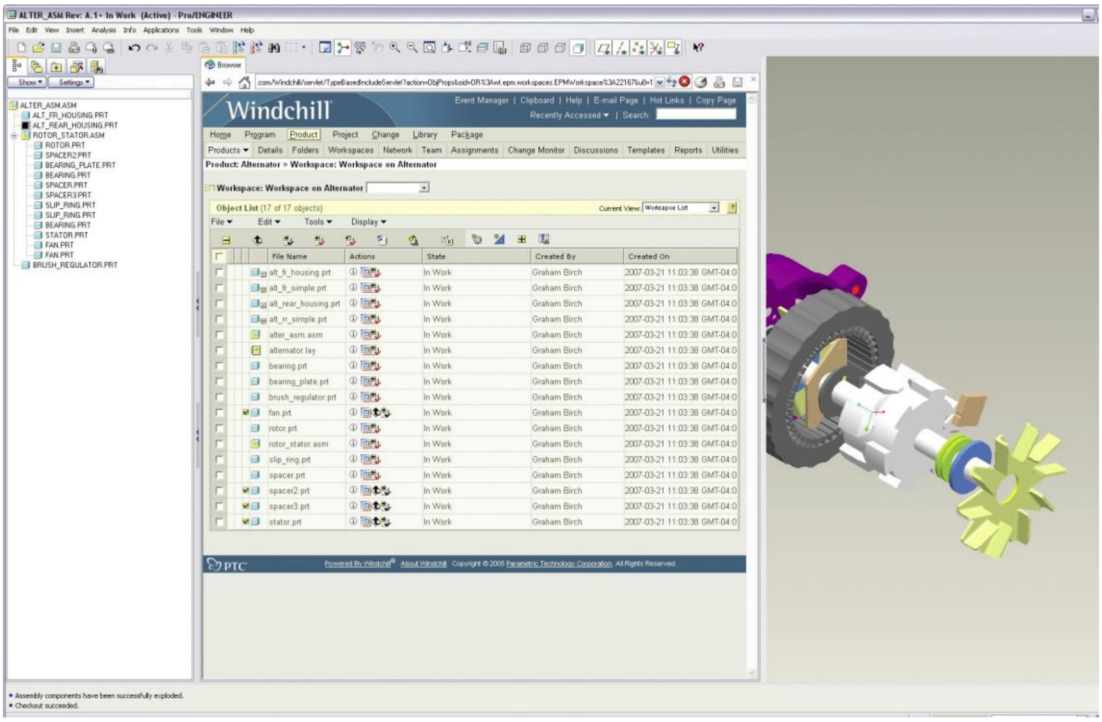
Product Database Management (PDM)



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Product Database Management (PDM)

PDM systems provide graphical visualization of the data with the following features:

♦ displaying the 3D data with customizable views (using rotation, zoom, pan functions)

♦ the assemblies can be exploded to view the individual components

♦ hide/show components

♦ customizable cross sections

♦ measurements on the 3D model

♦ remarking with geometry attached notes

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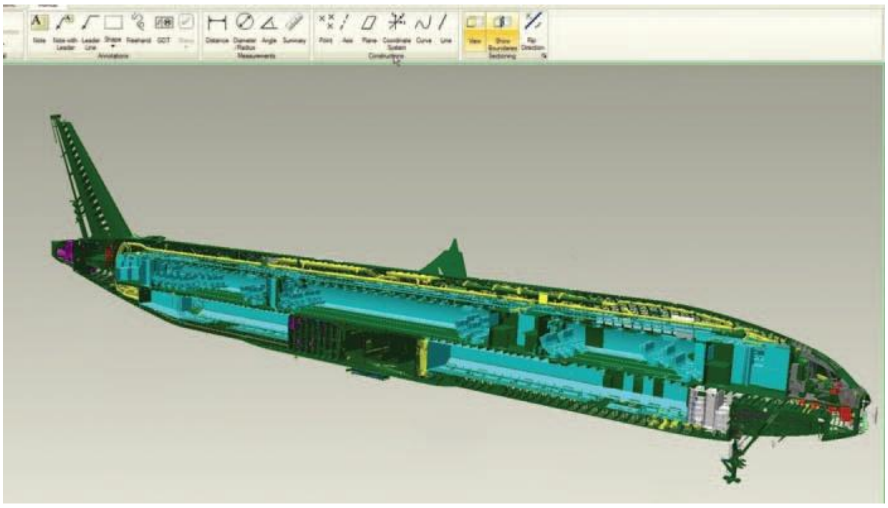
Product Database Management (PDM)

Visualization of a high complexity product in a web based UI



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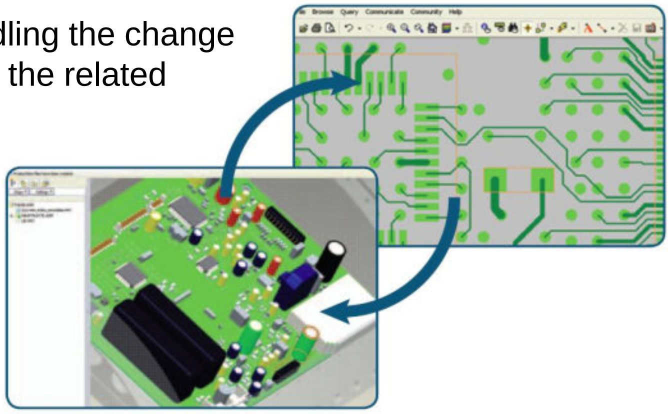
Product Database Management (PDM)

Tracking the engineering changes is one of the most important function of the PDM systems. This function encloses the following features:

♦ **version tracking**: storing the product versions and iterations, option to restore a previous version

♦ **log function**: logging the data access and change and other activities

♦ **change management**: handling the change requests and notifications and the related approver workflows



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Product Database Management (PDM)

The PDM systems support, an important element of the concurrent engineering, the teamwork. This kind of groupware functions are:

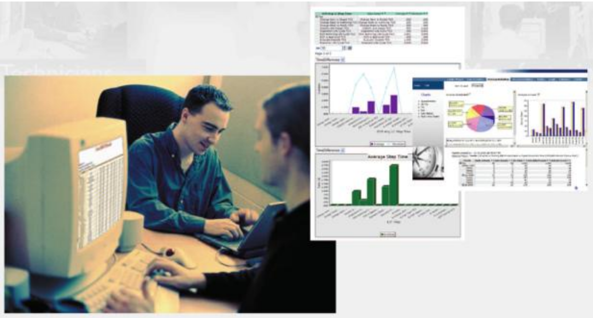
♦ unique access for modifying a specific data in the DB

♦ access rights for avoiding the unauthorized data accesses

♦ e-mail notification about the DB actions (modification, approval of a version, creation of a new object, etc.)

♦ supporting the electronic signatures

♦ other decision supporting functions (data collecting and visualization, making of statistics, reports, etc.)



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Product Database Management (PDM)

The highest level control of the procedures in a PDM systems is the process management with the following features:

- ♦ definition of the participants and the roles of the processes
- ♦ tasklist definition (participants, activities, milestones and notifications)
- ♦ process automation with automatic running of the tasklist steps
- ♦ process trekking with visualization of the status and the decision points
- ♦ support of complex product lifecycles build from automated processes and special gate processes

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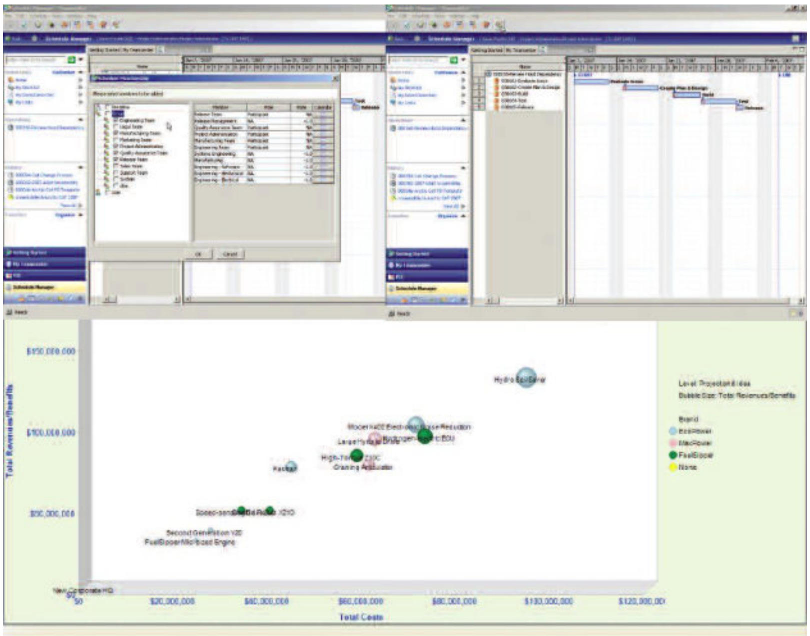
Product Database Management (PDM)

Derivative process data for teamwork support



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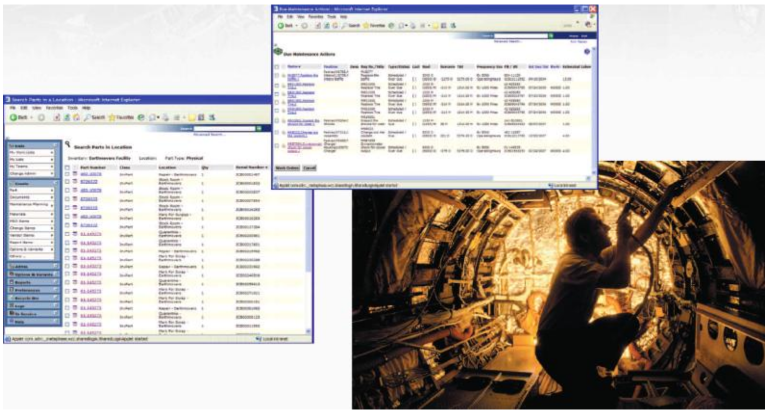
Product Lifecycle Management (PLM)

The PLM systems extend the PDM systems to the full lifecycle of the product. The PLM systems additionally control the following areas:

- ♦ prototype making
- ♦ manufacturing
- ♦ spare part manufacturing
- ♦ service



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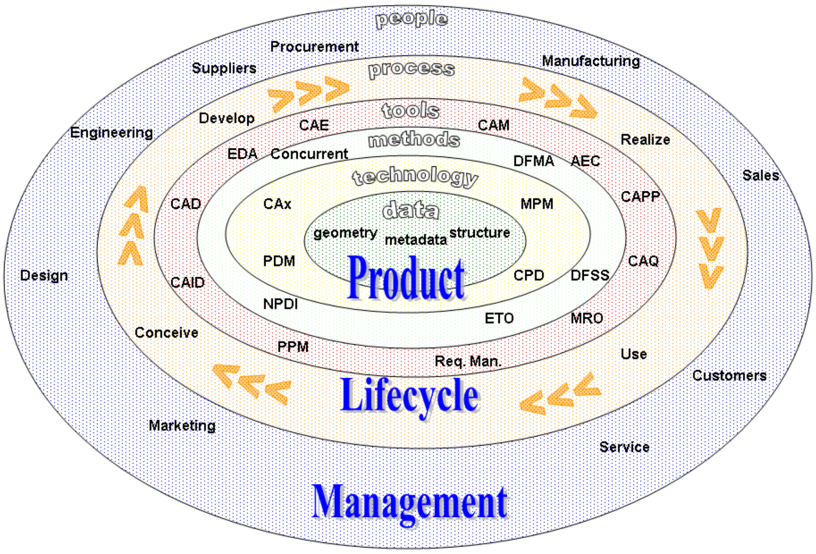
Product Lifecycle Management (PLM)

Areas of the PLM systems:



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Product Lifecycle Management (PLM)

The main advantages of the application of the PLM systems:

♦ significant decreasing in the time to market period

♦ better product quality

♦ lower prototype costs

♦ more accurate forecasts for time and costs

♦ easier identification of the market opportunities

♦ savings with recycling of the existing products

♦ framework for product optimization

♦ less reject and waste

♦ saving with integration of the overall design process

♦ help in creating documents for compliance with different standards

♦ data sharing with manufacturing subcontractors

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Product Lifecycle Management (PLM)

Additional specialized software modules of the PLM systems:

♦ **Systems Engineering, SE**
Process and system planing based on the consumer requirement.

♦ **Product and Portfolio Management, PPM**
The module monitors the running and suspended projects and helps the decision makers in the organization of the projects.

♦ **Product Design, CAx**
Different mechanical (MCAD) and electronic (ECAD) design software, simulation systems (FEA, CFD). These software provide the virtual testing opportunities of the product.

♦ **Manufacturing Process Management, MPM**
This module speeds up the manufacturing preprocessing and helps the optimization of the running manufacturing sequences.

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Product Lifecycle Management (PLM)

Application of the PLM systems in case of complex product:

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CAD BOOK

15. Peripheral technologies

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The product development process is not finished by the CAD work, the modelling and the drawing. The CAD model is suitable for other activities. This chapter presents some of them.
The first is the 3D scanning, when a real part will be converted to the CAD model by digitalization. There are several methods which will presented in the beginning. This technology is suitable for digital reproduction of a part.
The rapid prototyping technologies are able to create a real part based on the CAD model. The application of them has a wild range from the visualization to the functional test or real production.
One type of machine parts can be produced by cutting technologies (turning, milling). In order to productivity and accuracy CNC machine tools are applied, where the motion of the tools are controlled by the CNC programs. The CAM systems can be generated these programs based on CAD model of the part.

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15.1 3D SCANNING

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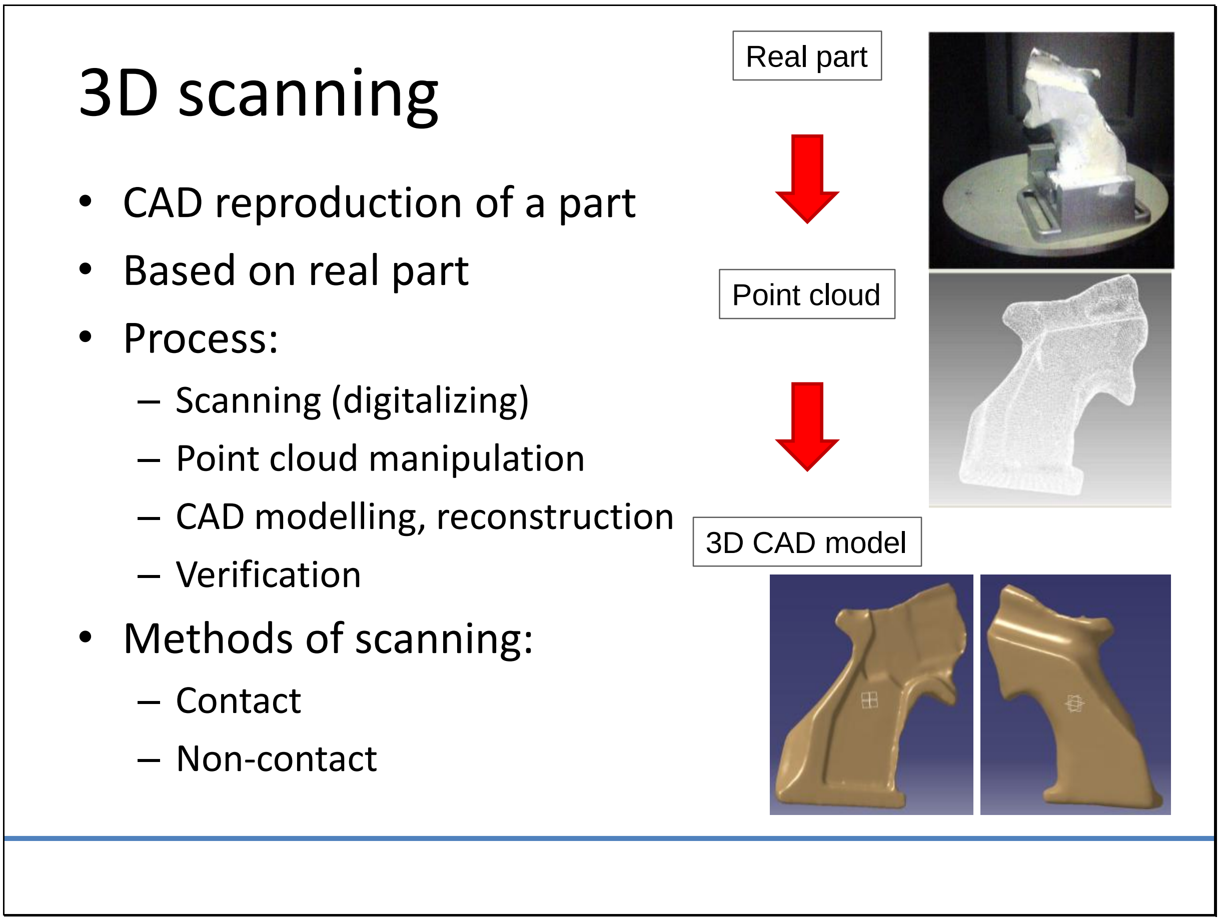

The aim of the 3D scanning is to digitalized a real part for additional CAx work. We use this methods, if there is no CAD documentation, no drawing documentation, or the shape of the part is too complicated to remodelling.

The scanning process consist of the next steps:

- Digitalizing of the shape by scanning machine,
- Manipulation of the point cloud, correct and smooth the points,
- Create 3D surfaces based on point cloud,
- Verification the surface, check the accuracy and correctness of the part.

For the scanning two type of machines can be applied: the contact and the non contact method.

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Contact scanning methods

- CMM
  - Rigid
  - Accurate
  - Programable

- Measuring arm
  - Flexible
  - Mobile
  - Manual

In case of contact method we use a coordinate measuring machine (CMM) or the measuring arm. These machines are the tools of geometrical measurement, but they are able to generate the appropriate data.

The CMM has a very rigid design, therefore the accuracy is high. The CMM in general a programmable device, so it is able to perform an automatic scanning, which is a big advantage, if we would like to create a dense point cloud.

The measuring arm is very flexible, we can use it in case of complicated parts. It is a mobile device, so we can digitalize a large parts however inside the laboratory. The measuring arm is manual use.

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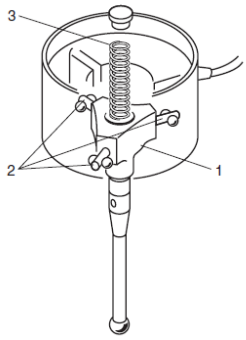
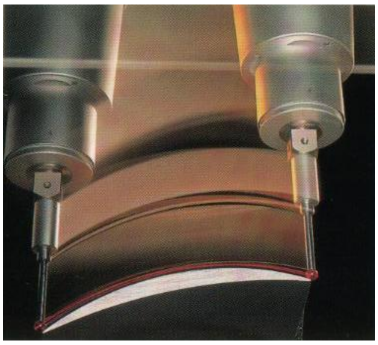
Contact scanning methods

- Touch-trigger head
  - Accurate
  - Small contact force
  - Slow process
  - Not expensive
- Analogue head
  - Accurate
  - Larger contact force
  - Fast process
  - High density of points





In case of use of coordinate measuring machine two type of head can be used.
The first is the touch-trigger head, which is very accurate, has a small contact force, but the scanning process is slow, because we have to touch the surface point by point. These devices are not too expensive.
The other type is the analogue head, which is accurate too, but in that case there is a larger contact force, because it can performed an automatic continuous scanning. The scanning process is faster and the density of the points will be high.

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Non-contact scanning

- No contact force
- Good for soft materials

- LASER scanning
  - Reflection problems
- CCD scanning
  - Expensive
  - Mobile

The second type of scanning methods is the non-contact method. There are no contact force, so we can scan soft materials too (like PU foam part).

For the scanning we can use laser beam. The reflection of the part can be problematic, therefore sometimes the surface need some preparation. In case of undercutted surfaces, we have to create and fit several point cloud patch.

The CCD camera based devices are portable, ensure very fast scanning, but very expensive.

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15.2 RAPID PROTOTYPING

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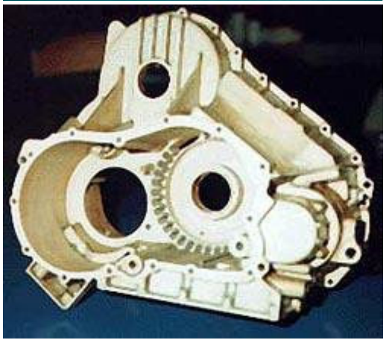
Role of rapid prototyping

- Presentation
- Pattern for casting
- Rapid tooling
- Rapid production



Rapid prototyping is the automatic construction of physical objects using additive manufacturing technology based on CAD model.

The aim of rapid prototyping methods is to create a prototype part directly from the CAD model in shorter time like in case of conventional methods.
These prototypes are suitable for presentation, we can use them as pattern for sand casting, the „prototype” can be a tool, mould or die, so we can use as production equipments, or in some case the prototype equal the final part, so we can use the RP methods for production.

The word "rapid" is relative: construction of a model with contemporary methods can take from several hours to several days, depending on the method used and the size and complexity of the model. Additive systems for rapid prototyping can typically produce models in a few hours, although it can vary widely depending on the type of machine being used and the size and number of models being produced simultaneously.

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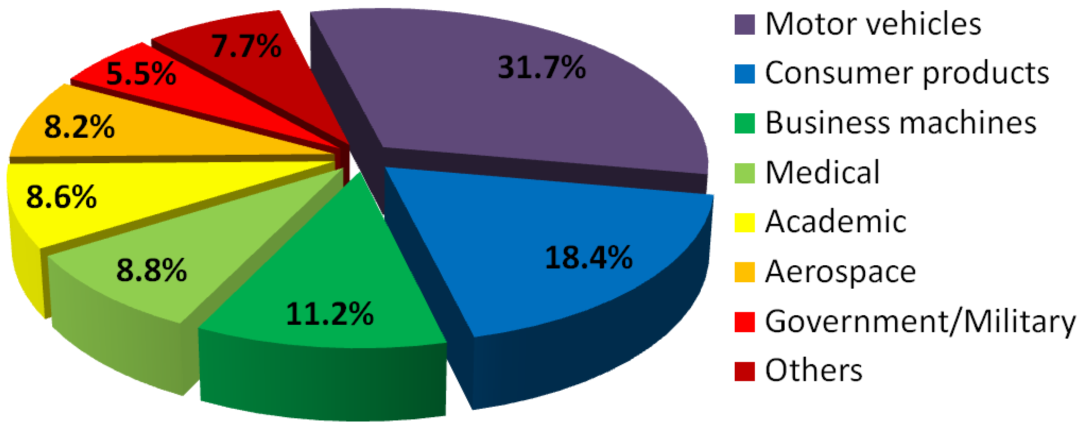
Application fields



The rapid prototyping has a great role in automotive industry and consumer products development, because the use of RP reduce the time and cost of development.

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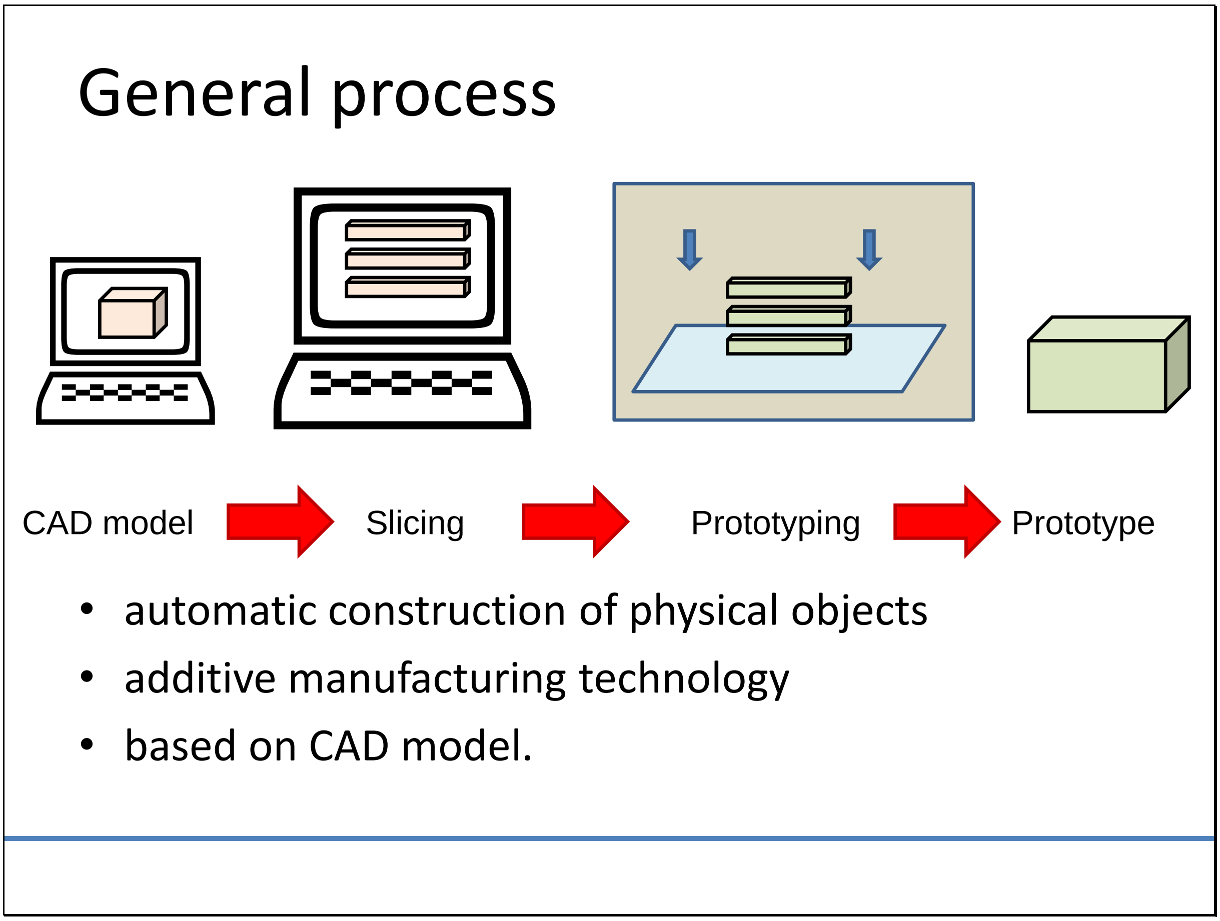

With additive manufacturing, the machine reads in data from a CAD drawing and lays down successive layers of liquid, powder, or sheet material, and in this way builds up the model from a series of cross sections. These layers, which correspond to the virtual cross section from the CAD model, are joined together or fused automatically to create the final shape. The primary advantage to additive fabrication is its ability to create almost any shape or geometric feature.

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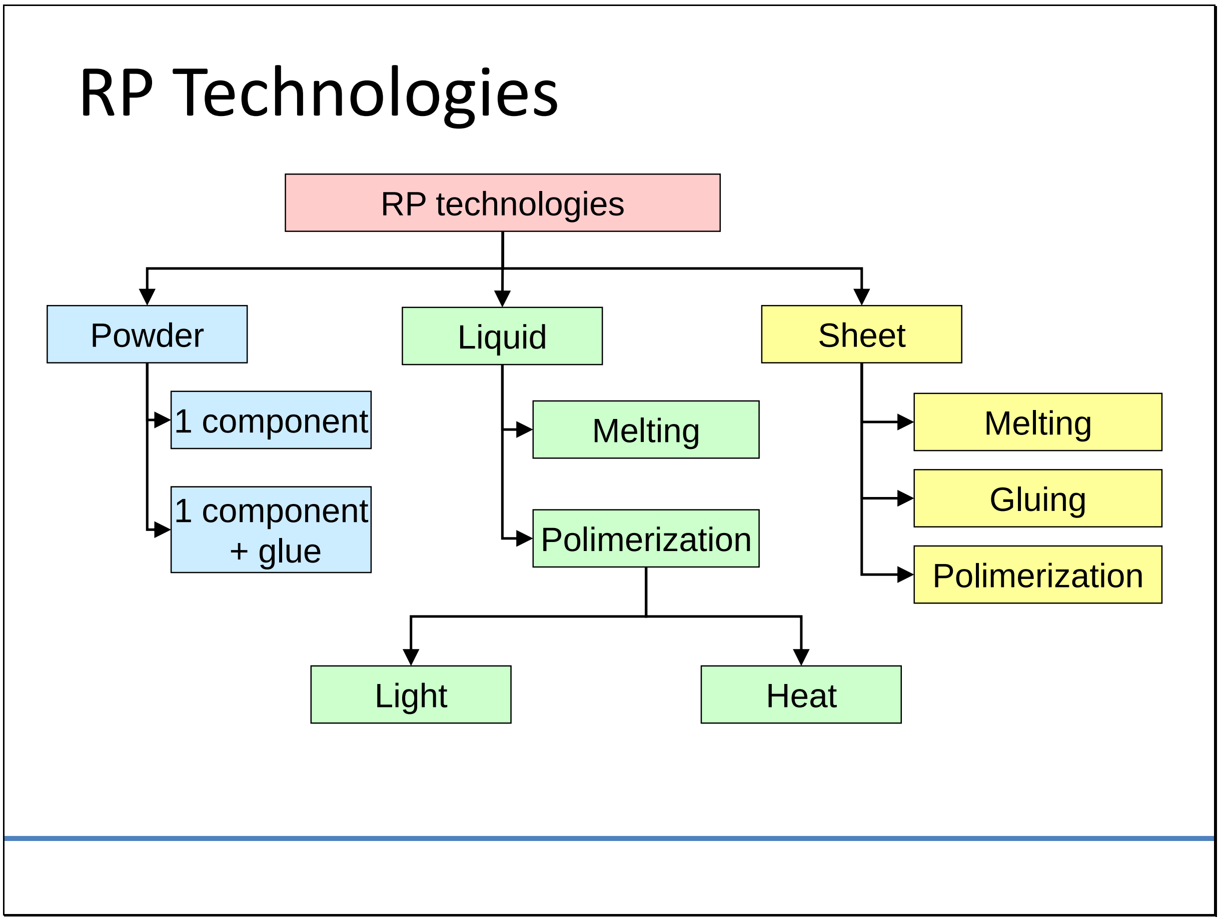

The methods classified by the type of raw material. The raw material can be a powder, and they will unite by melting or glue. We can use liquid raw material with polymerization or melting. The polymerization can catalyze with light or heat. Some methods use sheets typically by gluing.

Let's see the details of these technologies.

The most known RP technologies are the follows:

- SLA – Stereolithography
- SLS – Selective laser sintering
- LOM – Laminated object manufacturing
- FDM – Fused deposition modelling
- 3DP – 3D printing
- PolyJet
- SGC – Solid ground curing

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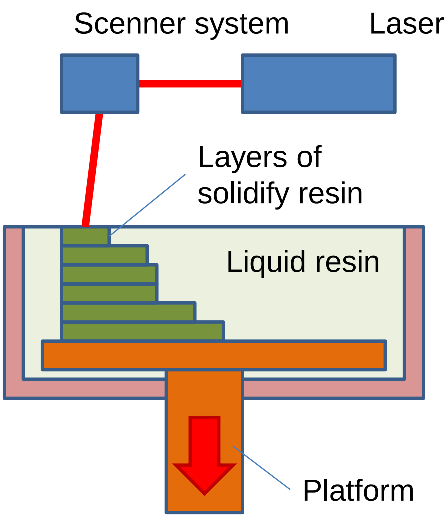
SLA - Stereolithography



- Layer thickness: 0,1 mm
- UV post processing required
- Strength: 30 MPa
- PBT, PA, PP, PC

Stereolithography is an additive manufacturing process using a vat of liquid UV-curable photopolymer resin and a UV laser to build parts a layer at a time. On each layer, the laser beam traces a part cross-section pattern on the surface of the liquid resin. Exposure to the UV laser light cures, solidifies the pattern traced on the resin and adheres it to the layer below.

After a pattern has been traced, the SLA's elevator platform descends by a single layer thickness, typically 0.05 mm to 0.15 mm. Then, a resin-filled blade sweeps across the part cross section, re-coating it with fresh material. On this new liquid surface, the subsequent layer pattern is traced, adhering to the previous layer. A complete 3-D part is formed by this process. After building, parts are cleaned of excess resin by immersion in a chemical bath and then cured in a UV oven.

Stereolithography requires the use of support structures to attach the part to the elevator platform and to prevent certain geometry from not only deflecting due to gravity, but to also accurately hold the 2-D cross sections in place such that they resist lateral pressure from the re-coater blade. Supports are generated automatically during the preparation of 3-D CAD models for use on the stereolithography machine, although they may be manipulated manually. Supports must be removed from the finished product manually.

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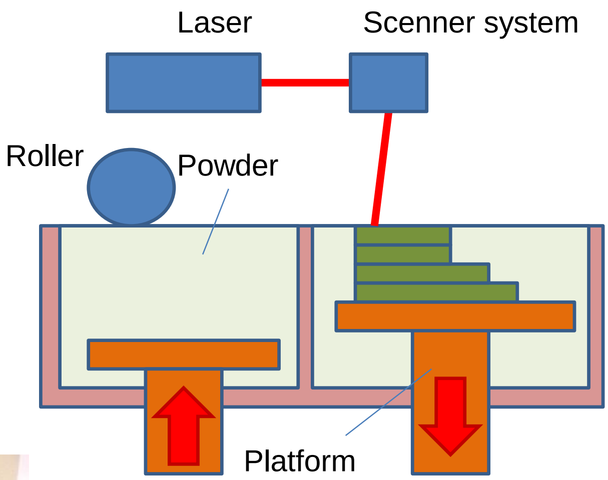
SLS – Selective laser sintering

- Wide variety of build material (plastic and metal)
- No support material
- No post processing
- Multiple part building
- High strength
- Thin wall
- Small parts
- Porous part



Selective laser sintering (SLS) is an additive manufacturing technique that uses a high power laser (for example, a carbon dioxide laser) to fuse small particles of plastic, metal (direct metal laser sintering), ceramic, or glass powders into a mass that has a desired 3-dimensional shape. The laser selectively fuses powdered material by scanning cross-sections generated from a 3-D digital description of the part (for example from a CAD file or scan data) on the surface of a powder bed. After each cross-section is scanned, the powder bed is lowered by one layer thickness, a new layer of material is applied on top, and the process is repeated until the part is completed.

Because finished part density depends on peak laser power, rather than laser duration, a SLS machine typically uses a pulsed laser. The SLS machine preheats the bulk powder material in the powder bed somewhat below its melting point, to make it easier for the laser to raise the temperature of the selected regions the rest of the way to the melting point.

Some SLS machines use single-component powder, such as direct metal laser sintering. However, most SLS machines use two-component powders, typically either coated powder or a powder mixture. In single-component powders, the laser melts only the outer surface of the particles (surface melting), fusing the solid non-melted cores to each other and to the previous layer.

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Compared to other methods of additive manufacturing, SLS can produce parts from a relatively wide range of commercially available powder materials. These include polymers such as nylon, (neat, glass-filled or with other fillers) or polystyrene, metals including steel, titanium, alloy mixtures, and composites and green sand. The physical process can be full melting, partial melting, or liquid-phase sintering. And, depending on the material, up to 100% density can be achieved with material properties comparable to those from conventional manufacturing methods. In many cases large numbers of parts can be packed within the powder bed, allowing very high productivity.

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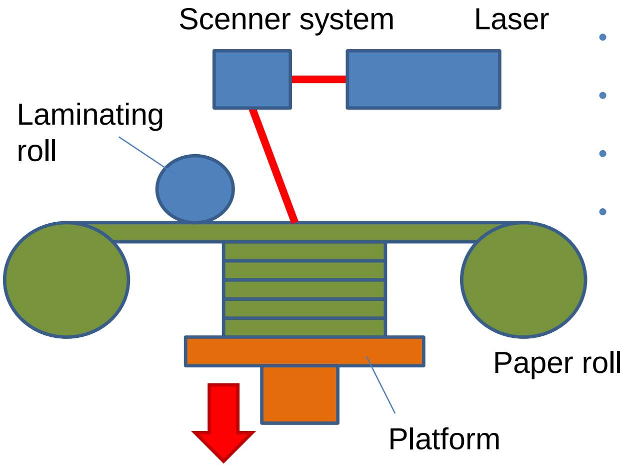
LOM – Laminated object manufacturing



- Paper model (thickness 0,1 mm)
- Cut by laser and laminate
- Painable
- Close cavity is impoossible

Laminated object manufacturing (LOM) is a rapid prototyping system developed by Helisys Inc. In it, layers of adhesive-coated paper, plastic, or metal laminates are successively glued together and cut to shape with a knife or laser cutter.

The process is performed as follows:

1.Sheet is adhered to a substrate with a heated roller.
2.Laser traces desired dimensions of prototype.
3.Laser cross hatches non-part area to facilitate waste removal.
4.Platform with completed layer moves down out of the way.
5.Fresh sheet of material is rolled into position.
6.Platform moves up into position to receive next layer.
7.The process is repeated.

The most important properties of LOM:

- Low cost due to readily available raw material
- Paper models have wood like characteristics, and may be worked and finished accordingly

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- Dimensional accuracy is slightly less than that of Stereolithography and Selective laser sintering but no milling step is necessary.
- Relatively large parts may be made, because no chemical reaction is necessary.

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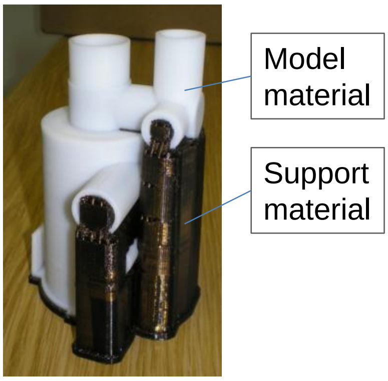
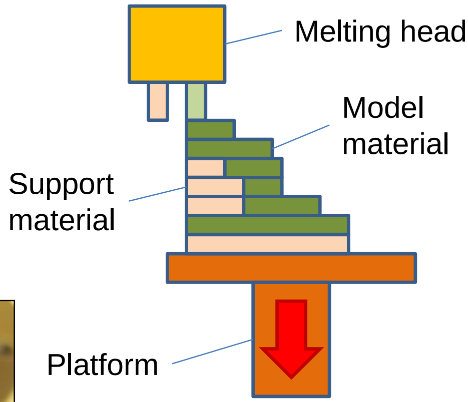
FDM – Fused deposition modeling

- ABS plastic wire
- Layer thickness: 0,2 mm
- No post processing
- Paintable, glueable
- Automatic support material processing





Fused deposition modelling (FDM) is an additive manufacturing technology commonly used for modelling, prototyping, and production applications. The technology was developed by S. Scott Crump in the late 1980s and was commercialized in 1990. FDM begins with a software process, developed by Stratasys, which processes an STL file (stereolithography file format) in minutes, mathematically slicing and orienting the model for the build process. If required, support structures are automatically generated. The machine dispenses two materials – one for the model and one for a disposable support structure.

FDM, a prominent form of rapid prototyping, is used for prototyping and rapid manufacturing. Rapid prototyping facilitates iterative testing, and for very short runs, rapid manufacturing can be a relatively inexpensive alternative.

Several materials are available with different trade-offs between strength and temperature properties. As well as acrylonitrile butadiene styrene (ABS) polymer, polycarbonates (PC), polycaprolactone (PCL), polyphenylsulfones (PS) and waxes.

The thermoplastics are liquefied and deposited by an extrusion head, which follows a tool-path defined by the CAD file. The materials are deposited in layers as fine as 0.125 mm (0.005") thick, and the part is built from the bottom up – one layer at a time.

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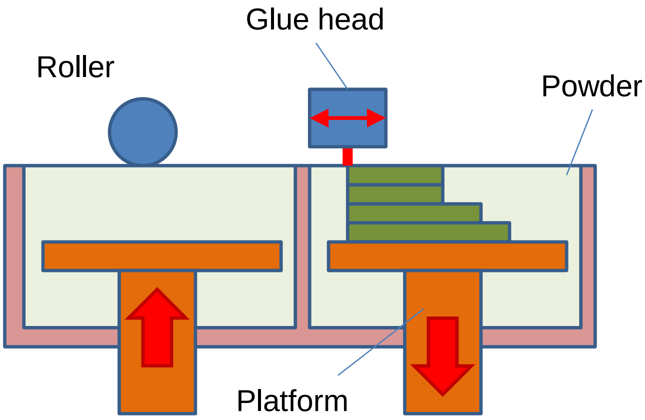
3DP – 3D printing

- Powder and glue
- Colour model
- Tempering post processing required



3D printing is a form of additive manufacturing technology where a three dimensional object is created by laying down successive layers of material. 3D printers are generally faster, more affordable and easier to use than other additive manufacturing technologies. 3D printers offer product developers the ability to print parts and assemblies made of several materials with different mechanical and physical properties in a single build process. Advanced 3D printing technologies yield models that can serve as product prototypes.

The method of 3D printing consists of an inkjet printing system. The printer creates the model one layer at a time by spreading a layer of powder (plaster, or resins) and inkjet printing a binder in the cross-section of the part. The process is repeated until every layer is printed. This technology is the only one that allows for the printing of full colour prototypes. This method also allows overhangs. It is also recognized as the fastest method.

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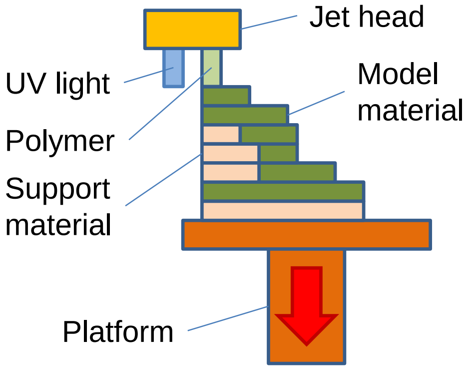
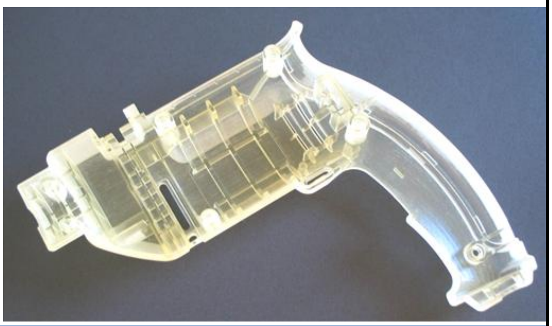
PolyJet



- Ink jet printing
- Direct polymer pronting
- Polymer + UV
- Multi component available
- Layer thickness: 0.02 mm !!!



PolyJet Matrix™ Technology, a new direction in 3D printing, is the first technology that enables simultaneous jetting of different types of model materials, available on Connex family of 3D Printing Systems.

Objet's patent-pending PolyJet Matrix technology works by jetting two distinct Objet FullCure® photopolymer model materials in preset combinations.
The dual-jet process can combine materials in several ways, enabling the simultaneous use of two different rigid materials, two flexible materials, one of each type, any combination with transparent material, or two jets of the same material.
Each material is funneled to a dedicated liquid system connected to the PolyJet Matrix block, which contains 8 printing heads. Two perfectly synchronized printing heads are designated for each material, including the support material.
PolyJet Matrix Technology controls every one of the 96 nozzles in every print head. Preset composites of model materials are jetted from designated nozzles according to location and model type, providing full control of the structure of the jetted material and hence of its mechanical properties. This enables each composite material, called a "Digital Material", to provide specific values for tensile strength, elongation to break, HDT and even Shore A values.

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The materials are jetted in ultra-thin layers onto a build tray, layer by layer, until the part is completed. Each photopolymer layer is cured by UV light immediately after it is jetted, producing fully cured models that can be handled and used immediately, without post-curing. The gel-like support material, which is specially designed to support complicated geometries, is easily removed by hand and water jetting.

PolyJet Matrix Advantages

- Enables the on-the-fly fabrication of Digital Materials™ – Composite materials that closely emulate the mechanical properties of the target design
- Enables the combination of black and white rigid materials, creating a wide range of greyscales suitable for consumer electronics and other applications
- Eliminates the need to design, print and glue together separate model parts made with different materials in order to create a complete model, saving printing and post-processing time
- Dramatically reduces the risk of error when creating complex moulds for double injection by enabling the majority of testing to be performed on early stage prototypes instead of requiring silicon moulds

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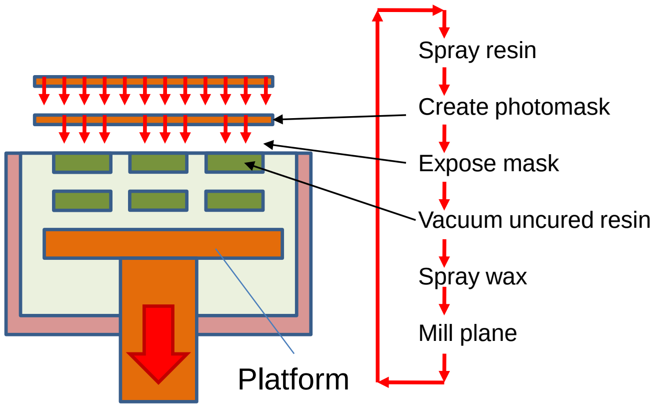
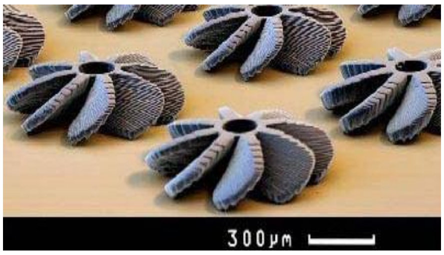
SGC – Solid ground curing



- Material: vax
- No support material
- Good surface quality

- Complicated
- Expensive



Solid Ground Curing (SGC), also known as the Solider Process, is a process that was invented and developed by Cubital Inc. of Israel. The overall process is illustrated in the figure above and the steps are illustrated below. The SGC process uses photosensitive resin hardened in layers as with the Stereolithography (SLA) process. However, in contrast to SLA, the SGC process is considered a high-throughput production process. The high throughput is achieved by hardening each layer of photosensitive resin at once. Many parts can be created at once because of the large work space and the fact that a milling step maintains vertical accuracy. The multi-part capability also allows quite large single parts (e.g. 500 × 500 × 350 mm / 20 × 20 × 14 in) to be fabricated. Wax replaces liquid resin in non-part areas with each layer so that model support is ensured.
The steps in the process are as follows.

First, a CAD model of the part is created and it is sliced into layers using Cubital's Data Front End® software. At the beginning of a layer creation step, the flat work surface is sprayed with photosensitive resin.

Next, the photomask is positioned over the work surface and a powerful UV lamp hardens the exposed photosensitive resin.

After the layer is cured, all uncured resin is vacuumed for recycling, leaving the hardened areas intact.

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The cured layer is passed beneath a strong linear UV lamp to fully cure it and to solidify any remnant particles.

In the fifth step, wax replaces the cavities left by vacuuming the liquid resin. The wax is hardened by cooling to provide continuous, solid support for the model as it is fabricated.

In the final step before the next layer, the wax/resin surface is milled flat to an accurate, reliable finish for the next layer.

Once all layers are completed, the wax is removed, and any finishing operations such as sanding, etc. can be performed. No post-cure is necessary.

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15.3 CAM SYSTEMS

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CAM system

- Computer Aided Manufacturing
- Tasks:
  - Cutting tool path design
  - Generation of NC program
- Elements of a CAM system:
  - Tool path generation
  - Edit of tool path
  - Optimization of tool path
  - Material and tool database
  - Machining time calculation
  - NC postprocessing

The aim of the CAM systems is to connect the virtual CAD model and the real manufacturing. The computer aided manufacturing (CAM) at first look seems a very complicated and ambitious term, but in real it means only the design of cutting tool path and the generation of NC program. The NC program is the alphanumerical code for control the work of CNC controlled automatic machine tools.

In case of complex part geometry the design of cutting tool path, which will generate the surface of the part, needs a lot of calculation, which takes lot of time. The CAM systems perform these calculations based on CAD model.

The typical parts of a CAM system are the next:

- Tool path generation
- Edit of tool path
- Optimization of tool path
- Material and tool database
- Machining time calculation
- NC postprocessing

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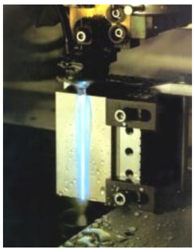
CAM classification

- Manufacturing technology
  - Milling
  - Turning
  - Cutting (laser, water jet, oxyfuel cutting, plasma arc, wire-edm)
  - Coordinate measuring machine
- Degree of freedom








The CAM systems can be classified by several viewpoints.
The first is the type of the machine tool or the applied cutting technology. So we can talk about CAM systems for milling, turning, cutting (laser, water jet, oxyfuel cutting, plasma arc, wire-edm) technologies or coordinate measuring.
The other classification based on the degree of freedom. The degree of freedom is the one axis moving possibility. The degree of freedom depends the technology and the design of the machine tool. Let us see some example.

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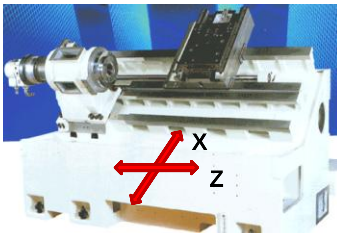
Degree of freedom in CAM systems

- Number of direction of moving possibilities
- 1D – moving in one axis
  - Drilling
- 2D – parallel moving in two axis
  - Turning
  - Cutting (laser, plasma, water, wire-edm)
- 2.5D – machining in x-y plane + moving for depth
  - Some milling operation: face milling, roughing, z-level milling



In case of 1D the tool has only one moving direction, like drilling.
In case of 2D the tool can move parallel in two direction. The turning and some cutting technology is typically 2D manufacturing.
The 2.5D means, that the 2D parallel moving possibilities is completed by a 3rd step motion, like some milling process.

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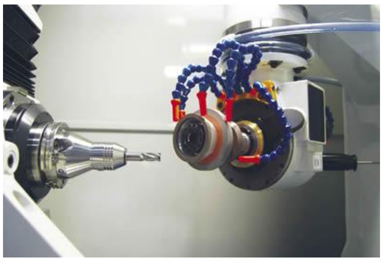
- 3D – parallel moving in three axis
  - Finishing milling of free form surfaces
  - Coordinate measuring machine
- 4D
  - Cutting in two parallel plane (eg. wire-edm)
  - Twin spindle lathe
- 5D
  - 5D milling: 3 linear axis + 2 rotary axis
- 6D
  - Industrial robots
- xD
  - Multi axis machine tools (eg. Tool grinding machine)



The 3D means simultaneous moving in 3 direction. The free form CNC milling and the coordinate measuring machine need this type of control.
The higher dimensions means complicated machine tools.
The 4D can be 2x2D, like in case of wire EDM of twin spindle late, or 4 simultaneous moving, like 3 linear and 1 rotational, or other.
The 5D is typically means 5D milling, when the 3 linear motion is competed by 2 rotational.
The industrial robots generally has a 6D control, like a humanoid robot, which has 6 rotary axes.
Some special machine tool required more axes.

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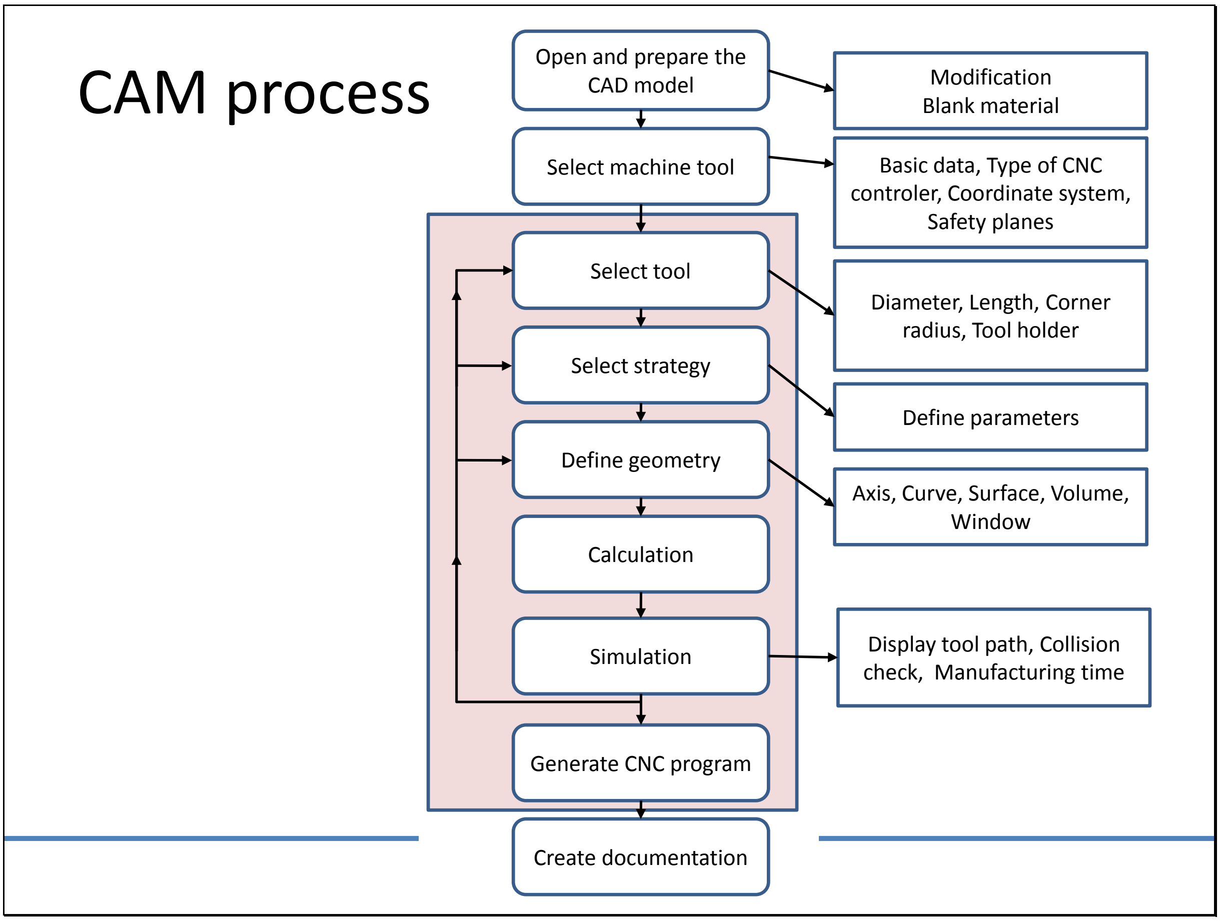

This pictures shows and summarized the typical workflow of a CAM system.
The order of the definition can be different in different CAM systems, but the definition of the listed data is required every case for appropriate work.
The most important application of the CAM systems is the milling technology, so hereafter I focus to this application.

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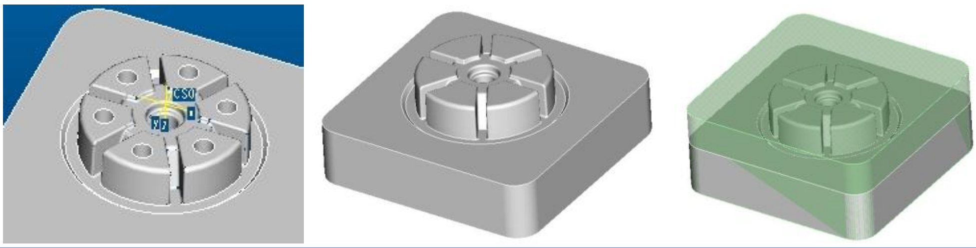
0. – Manufacturing process planning

1. – Open and prepare the CAD model

- File formats, data loss
  - Native / Neutral (dxf, step, iges, vda, stl etc.)
  - Parametric / Model hystory / Surface bugs
- Modification of the CAD geometry
- Surface pathing (eg. holes)
- Define blank material



The CAM workflow start with the manufacturing process planning. We have to plan the manufacturing process, because during the CAM workflow we take only definition. The CAM system can not create a process, can not select tool, can not define cutting parameters. The engineer will define them.
First, the CAD model is opened in the CAM system in native or neutral formats depends on CAM systems and consider the advantages and disadvantages. More details about data exchange is described in 13nd chapter.
If need and possible the geometry is corrected of modified, and the blank material geometry is defined.

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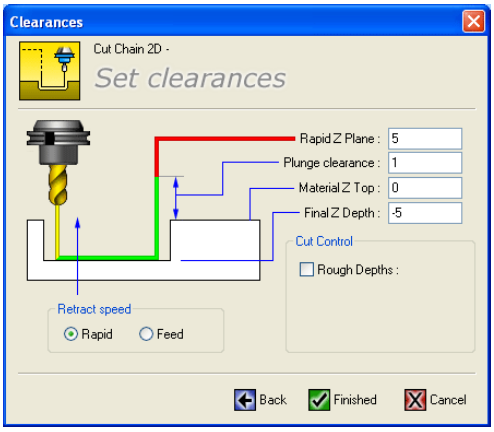
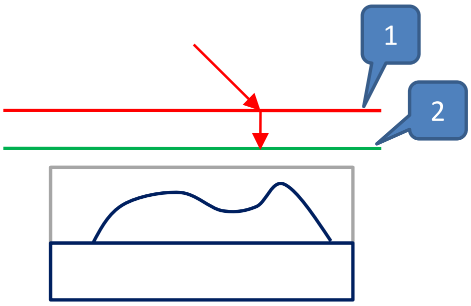
2. – Selection of machine tool

- Basic data (workspace, limits of cutting parameters etc.)
- Type of CNC controller
- Define coordinate system
- Define safety planes





1 – *safety plane*, over this plane the 3D rapid motion enabled
2 – retract plane, plane of connection motions

The second step is the machine tool selection. Here we have to define some basic data of the machine tool, like the size of the workplace, the limits of cutting parameters. We have to select the type of CNC controller, because it is an essential data for generate NC program.
The next step is the definition the coordinate system. The coordinate system gives the null point of the program.
In lot of CAM system we can define 2 planes over the part. Over the safety plane the 3D rapid motion is enabled, under it, the tool can be move fast only in perpendicular to the plane. The retract plane defines the level of the connection motions. Sometimes these two planes are same.

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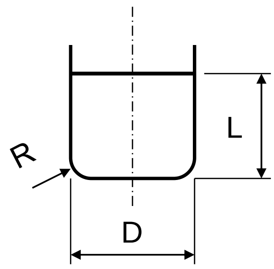
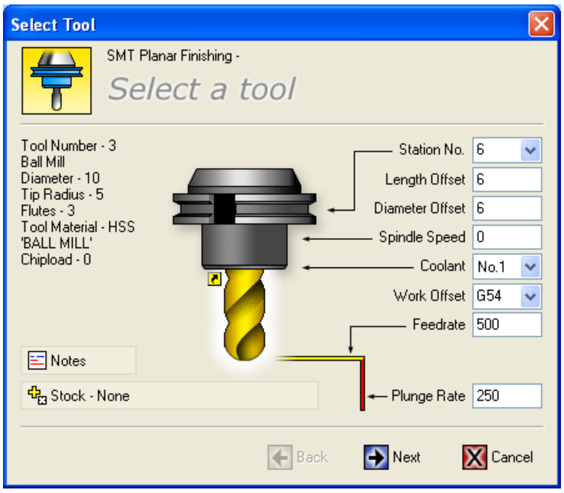
3. – Selection of cutting tool

- Tool data base
- Basic geometrical data
  - Diameter – D
  - Length - L
  - Corner radius - R
- Cutting parameters:
  - Material
  - Roughing / Finishing
  - n (v_c), v_f





The third step is the cutting tool selection or define. Lot of CAM systems contains a tool database, which contains geometric descriptions of cutting tools. This database contains the tools, which are exist in the machining workshop. The CAM system need only three geometric parameters of cutting tools in case of milling: diameter, length and corner radius. In general the database contains the recommended cutting data.

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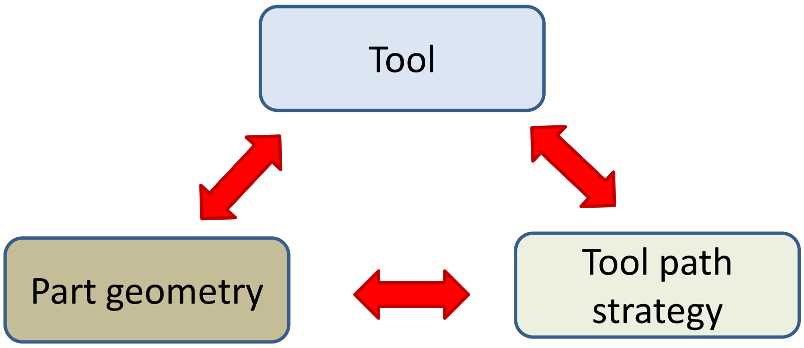
4. – Selection of tool path strategy

- Technology oriented tool path strategies
- Standard strategies + CAM system oriented special strategies
- One task – more strategies
- Tool – Part geometry – Tool path strategy



The most important part of a CAM system is the list of manufacturing strategies.
The tool path strategy define the character of the machining. The CAM engineer have to select the most appropriate strategy, consider the part geometry and the cutting tool.
The CAM systems contains „standard” strategies and CAM system oriented special strategies.
One manufacturing task can be solved by several different ways, the CAM systems ensure lot of tools for successful manufacturing.

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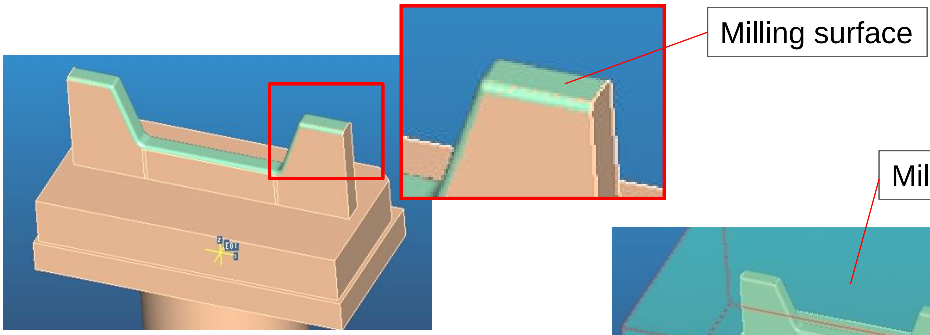
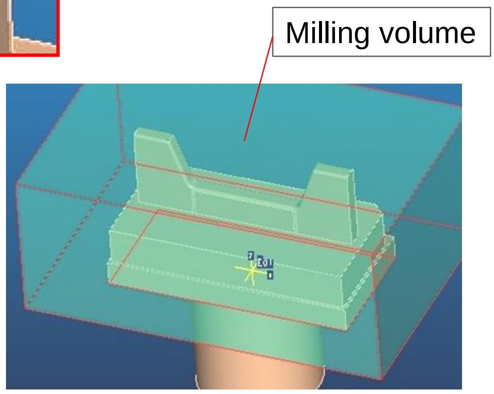
5. – Selection of machined geometry

- Axis, Curve, Surface, Volume
- On the CAD model / Associative definition





6. – Calculation of tool paths

In the next step we have to select or defined the geometry related to the selected strategy. We can select different type of geometrical elements:

- Axis – for hole making,
- Curve – engraving, slot milling,
- Surface – surface milling,
- Volume – rough milling.

We can select the existing geometry elements of the CAD model, or we can define new elements.

After the definition the CAM system performs the calculations. Typical it takes less the 1 minute.

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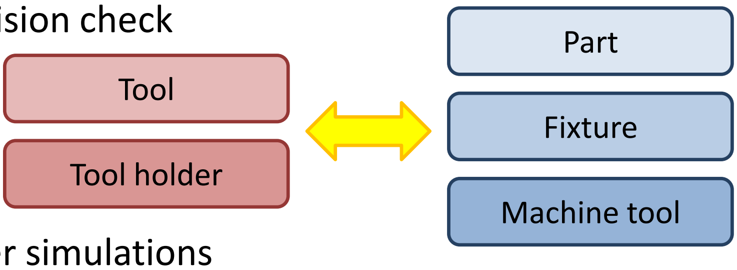
7. – Simulation

- Display the tool path
- Manufacturing simulation
  - Part + Tool
  - + Machnie tool
  - + Fixture
- Coolision check



- Other simulations
  - Machining time
  - Machining power

After the calculation we can control the result by simulations.
The simplest simulation is the simple display of the tool path.
The manufacturing simulation we can follow the machining process, we can see the generated part geometry. Basically this simulation contains only the part and the tool geometry, but we can complete the simulation with the machine tool and the fixture.
The other type of the simulation is the collision check, when the collision between the tool and the part, the tool and the fixture or machine tool, or the tool holder and other elements are detected.
The machining time calculation is an important type of the simulation. Some CAM systems are able to calculate more process parameter, like machining power.
The next pictures show some example about simulation.

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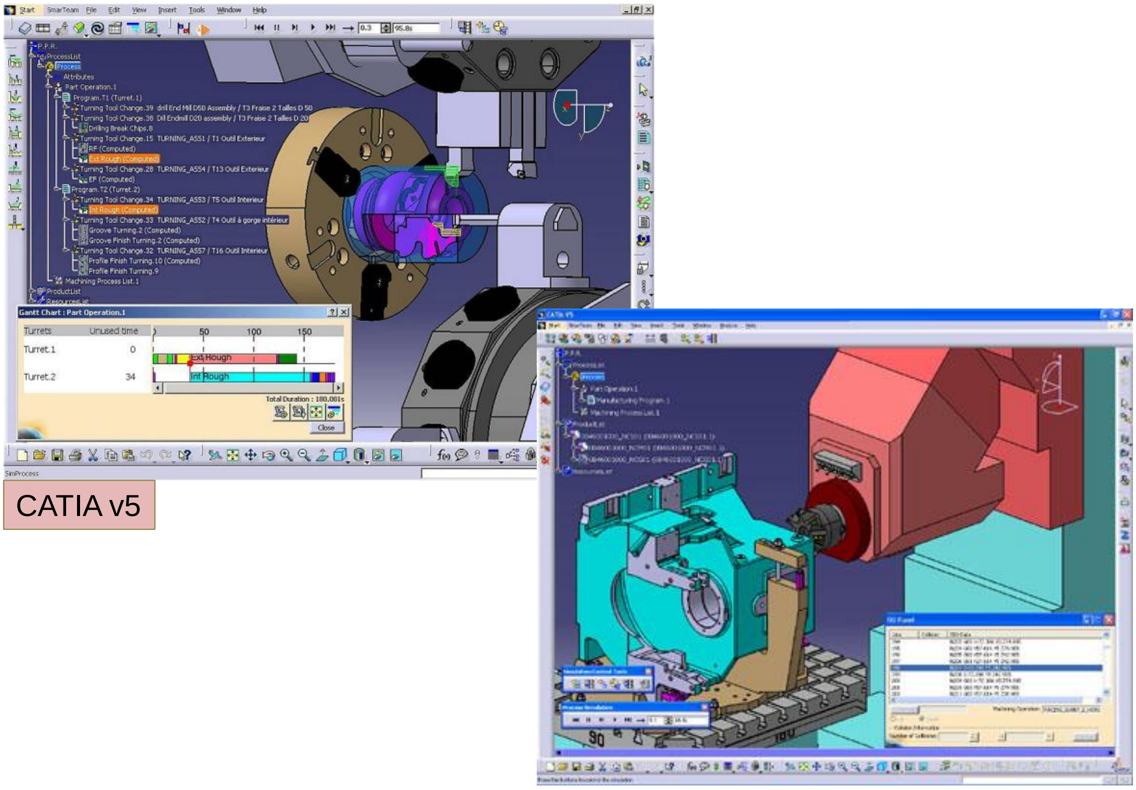

Simulation of turning and milling in CATIA V5 CAM system.

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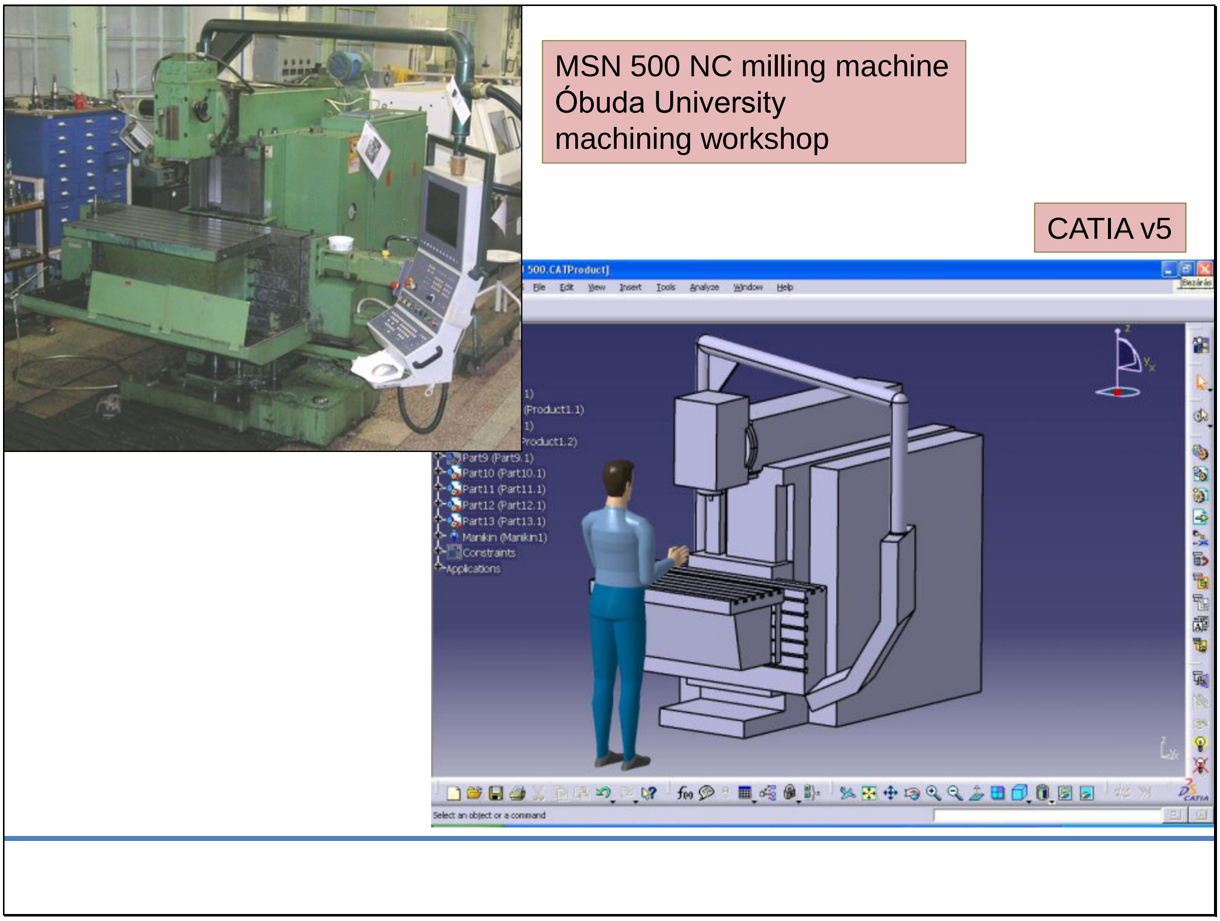

The simulation of the MSN500 NC milling machine in CATIA v5 CAM system.

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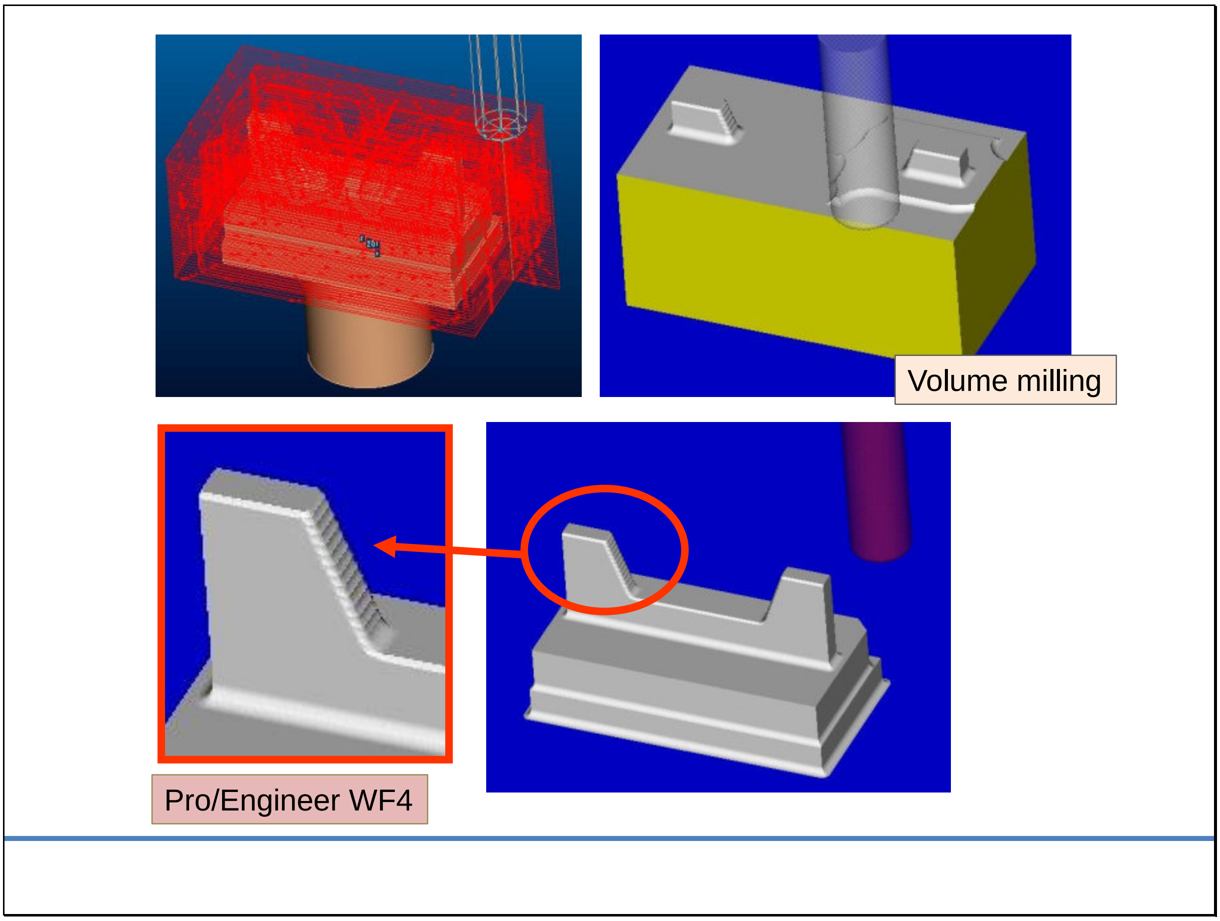

Machining simulation of milling of an EDM electrode in ProEngineer WF4.
The simulation shows the stepped surface after the roughing process.

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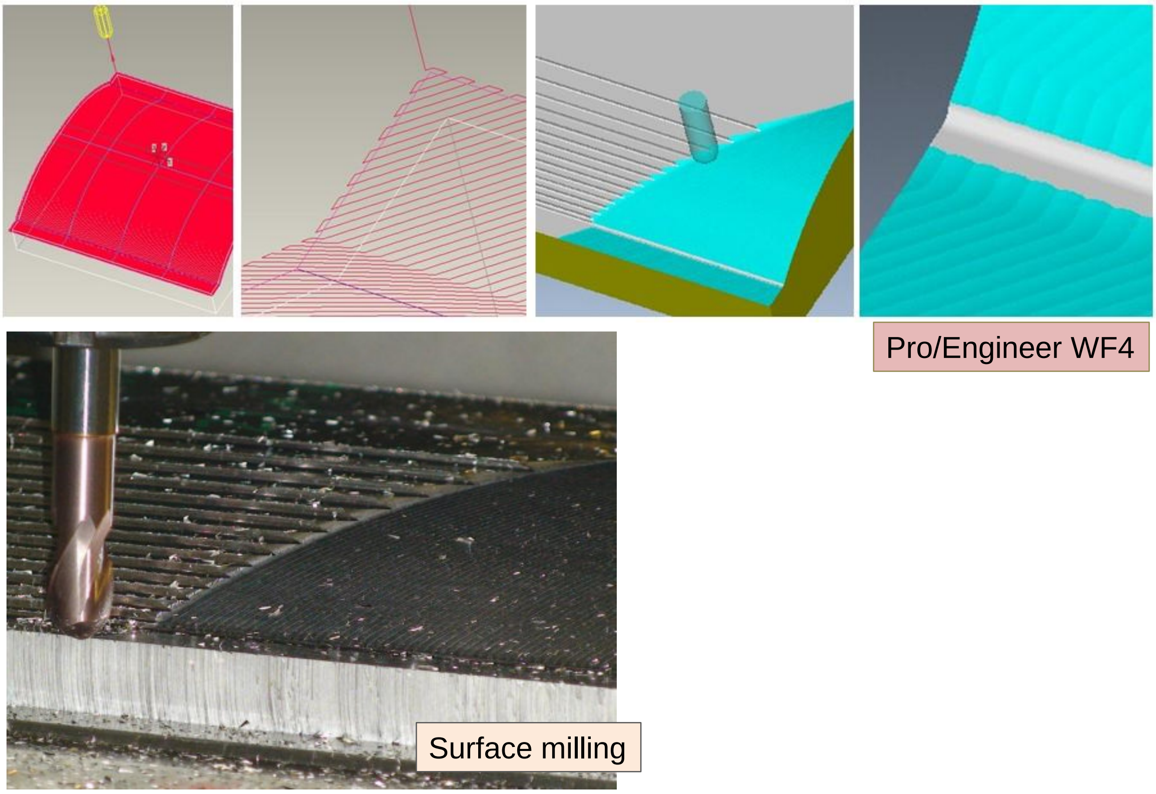

Simulation of surface milling in ProEngineer WF4.
The „tool path display” shows only the path, but the simulation displays the surface quality.

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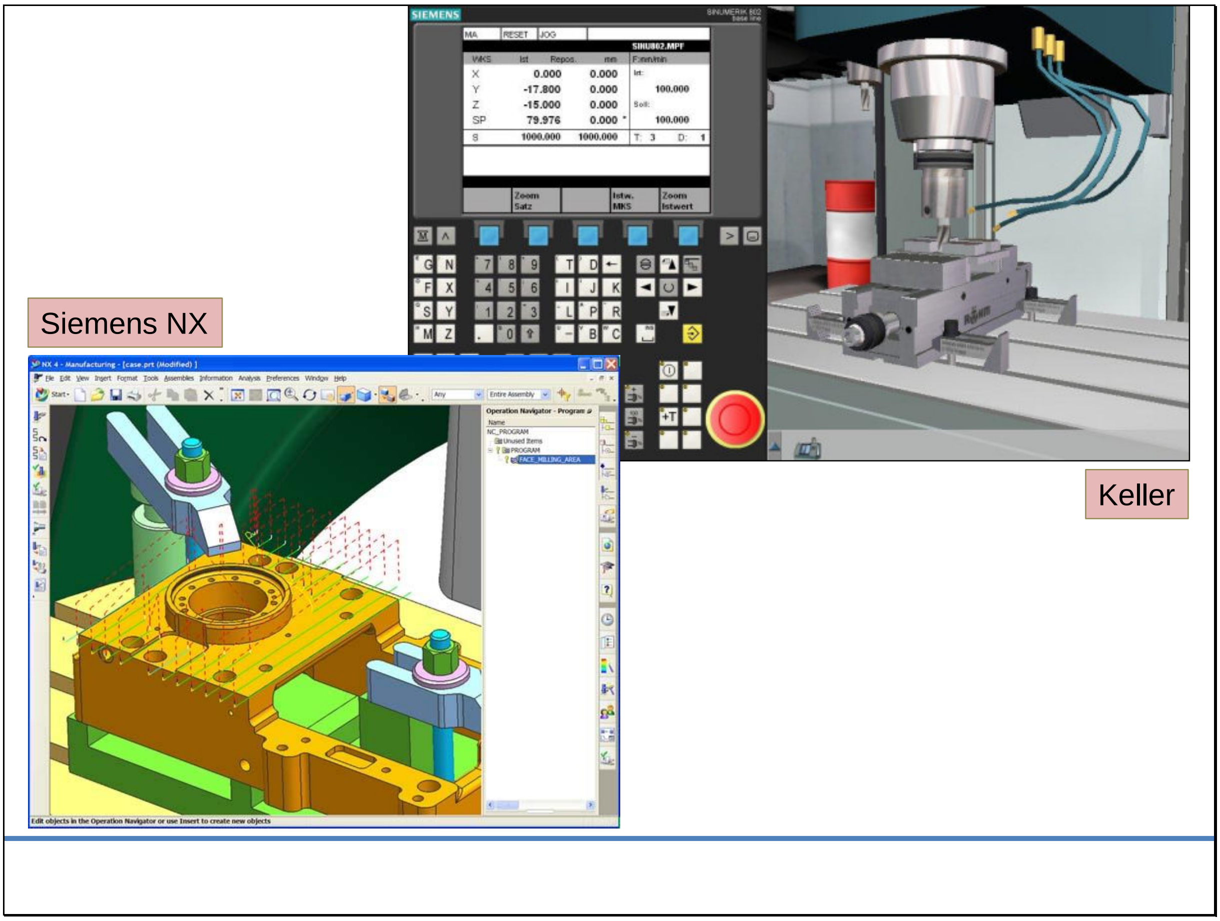

Some simulation system ensure the possibilities of simulation the complete manufacturing environment.

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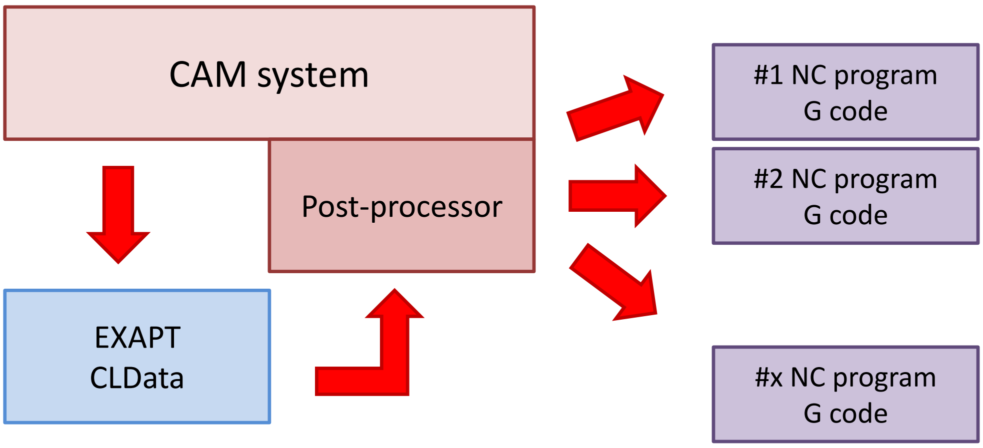
8. – Create NC program

- Independent format (APT / EXAPT)
- CNC controller oriented format (post-processing)



If the tool path is pass in the simulation, we can generate the NC code for machine tool. This process consists of two step. First we create an independent general code, and during the second step the general code is transformed to the CNC controller oriented format. However the format of NC code is standard, the different controller use a little bit different codes. This transformation is called postprocessing, and the software component is the postprocessor.

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9. – Documenting

- Location of the coordinate system
- Name of NC program
- Tool data
- Cutting parameters
- Manufacturing time

The last step is the documenting.
The CAM documentation contains all necessary data for production:

- Location of the coordinate system
- Name of NC program
- Tool data
- Cutting parameters
- Manufacturing time

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