# Zusammenfassung CAD Theorie

Diese Zusammenfassung haben Aaron und Benedict während der Vorbereitung auf die CAD Prüfung zusammengestellt. Sie beinhaltet vor allem die wichtigsten Slides aus den Vorlesungen von Prof. Dr. Kristina Shea aus dem Jahr 2018 und verweist auf das Technische Taschenbuch, Schaeffler, Aufl. 2017, welches Bestandteil der Hilfsmittel sein sollte, die du zur CAD Prüfung mitnimmst.
Da dies eine Kurzfassung der Vorlesungen ist, ist es empfehlenswert, bei Themen, die dir nicht direkt aus dem Skript erkenntlich sind, in den Vorlesungsunterlagen nachzusehen. Des Weiteren sind die wöchentlichen Übungen ein wichtiger Bestandteil der Prüfungsvorbereitung und sollten sorgfältig erledigt werden.
Für den praktischen Teil der Prüfung ist es zudem empfehlenswert, einen Überblick über die wichtigsten NX-11-Befehle ausgedruckt dabeizuhaben.

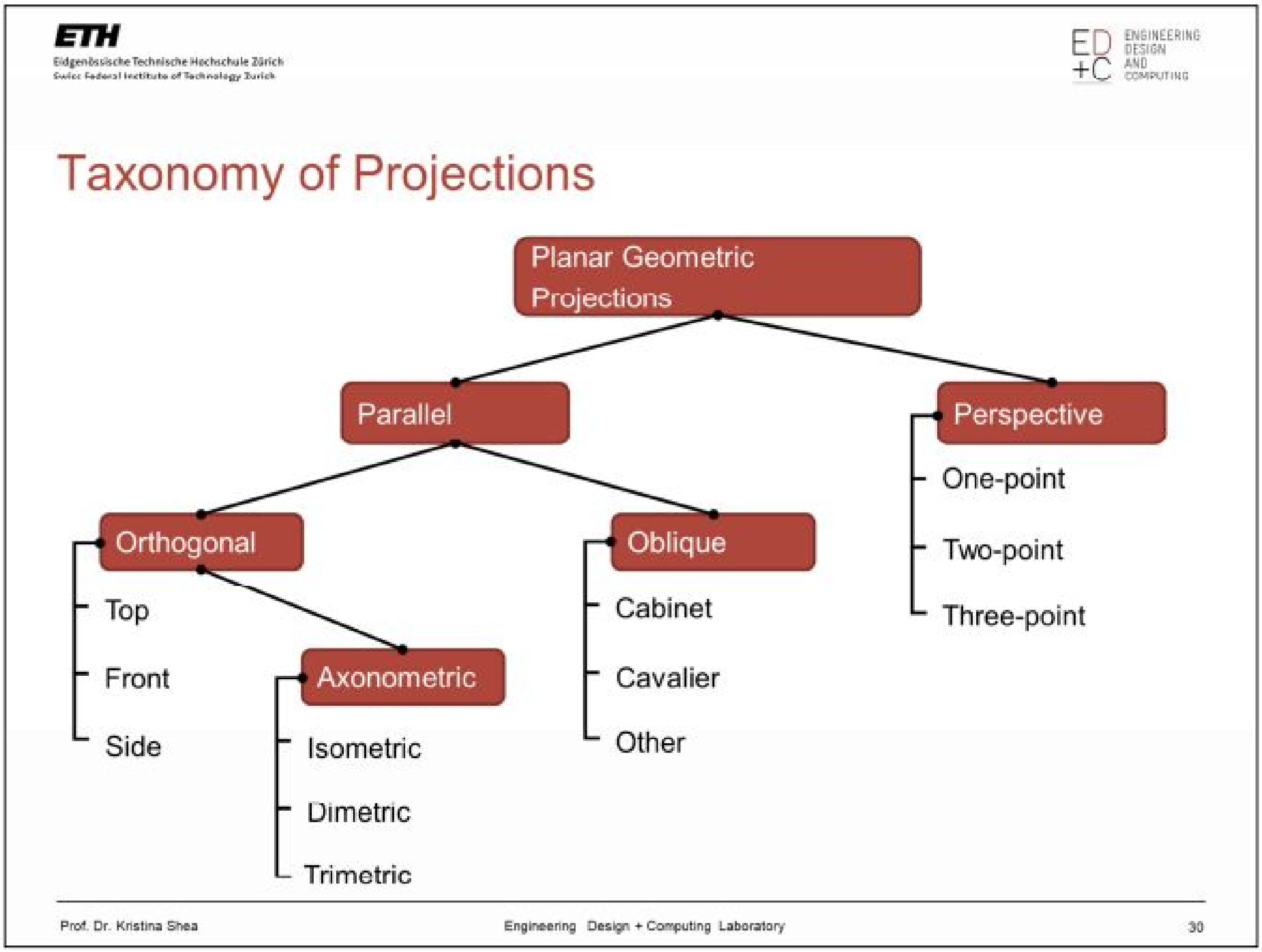
Ein Garantie der Vollständigkeit und Richtigkeit dieser Zusammenfassung ist nicht gegeben.

Viel Erfolg bei der Prüfung wünschen euch Aaron und Benedict.

# Inhaltsverzeichnis

# Projektionen und Ansichten

**Zusätzliche Informationenen: Technisches Taschenbuch, Schaeffler, Aufl. 2017 Seite 325 ff.**




**Parallel projection (*Parallelprojektion*)** - maps parallel lines in the space to parallel lines in the projection plane.

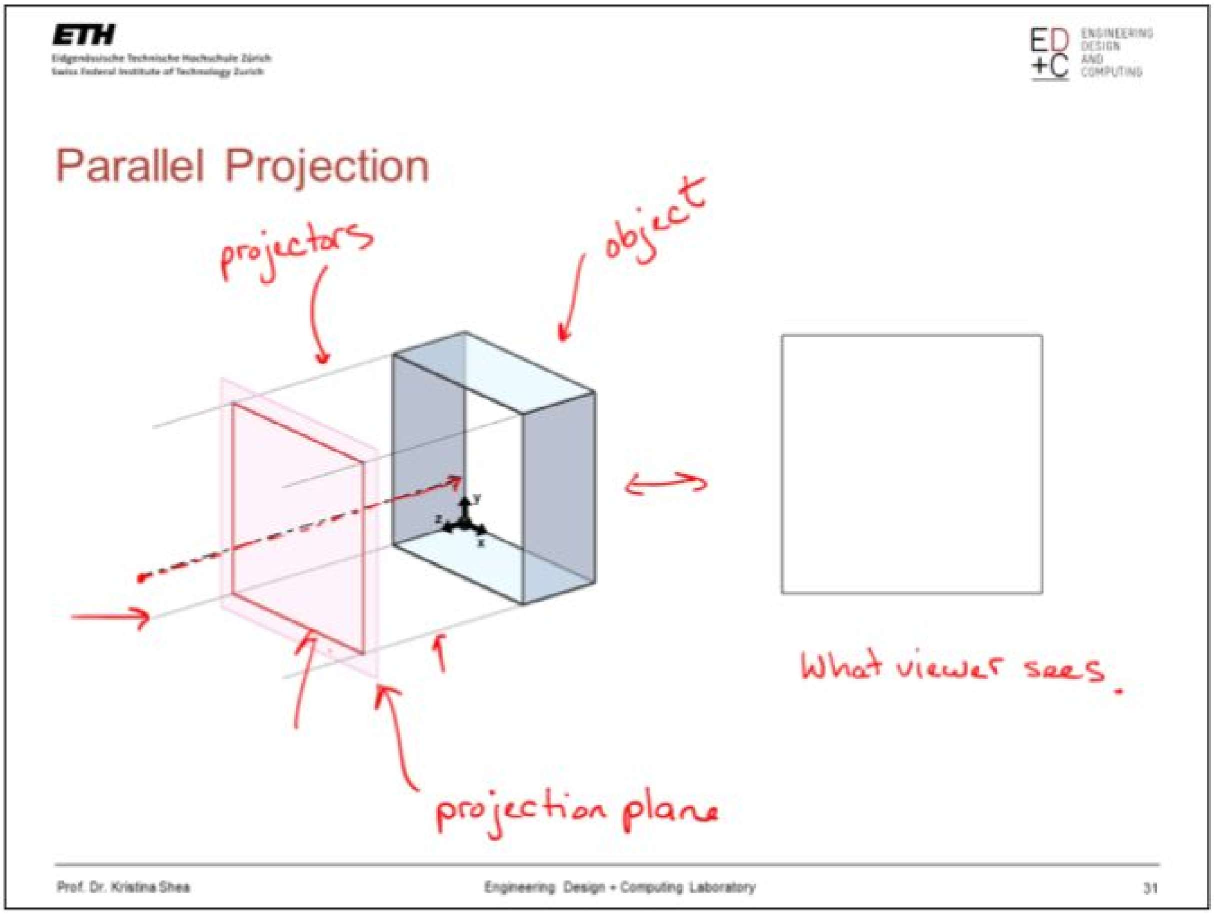
**Perspective projection (*Perspektivische Projektion*)** - maps parallel lines in the space to parallel lines in the projection plane if and only if the lines are parallel to the projection plane.

**Orthogonal projection (*Orthogonalprojektion*)** - parallel projection which maps perpendicular onto the projection plane.

**Oblique projection (*Schiefwinklige Projektion*)** - parallel projection which maps non-perpendicular onto projection plane.

**Axonometric projection (*Axonometrische Projektion*)** - orthogonal projection which tries to show all three sides of a 3D object.

**Parallelprojektion**



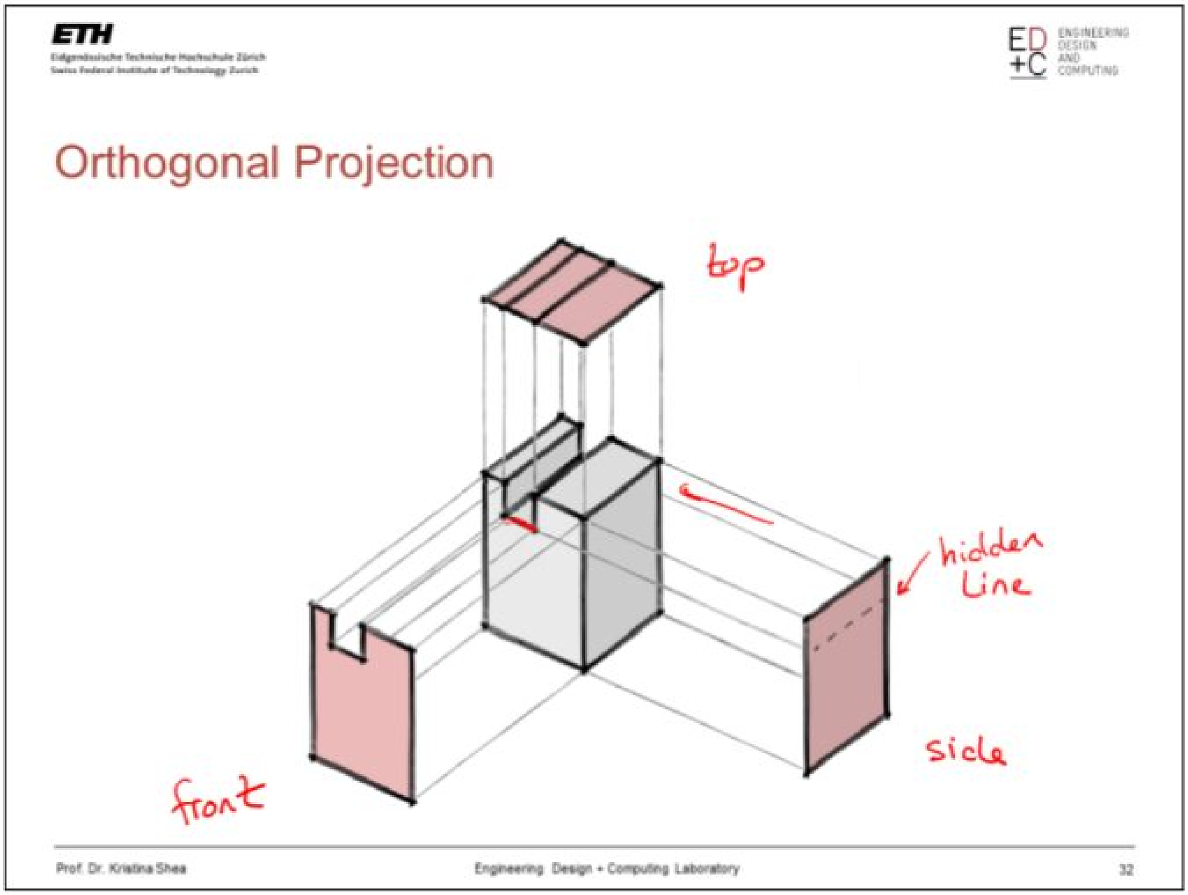

**Projectors (*Projektionslinie*)** – straight projection rays that start at the **center of projection (*Projektionszentrum*)**, pass through each point of the object and intersect the **projection plane (*Projektionsebene*)**.

**Parallel projection (*Parallelprojektion*)** – projectors are perpendicular to the projection plane, center of projection located at infinity (thus not shown in the image).

In general for parallel projections the projection plane may intersect more than one principle axis (*Hauptachse*).

The image shows a special case of parallel projection, the orthogonal projection.

**Orthogonalprojektion**

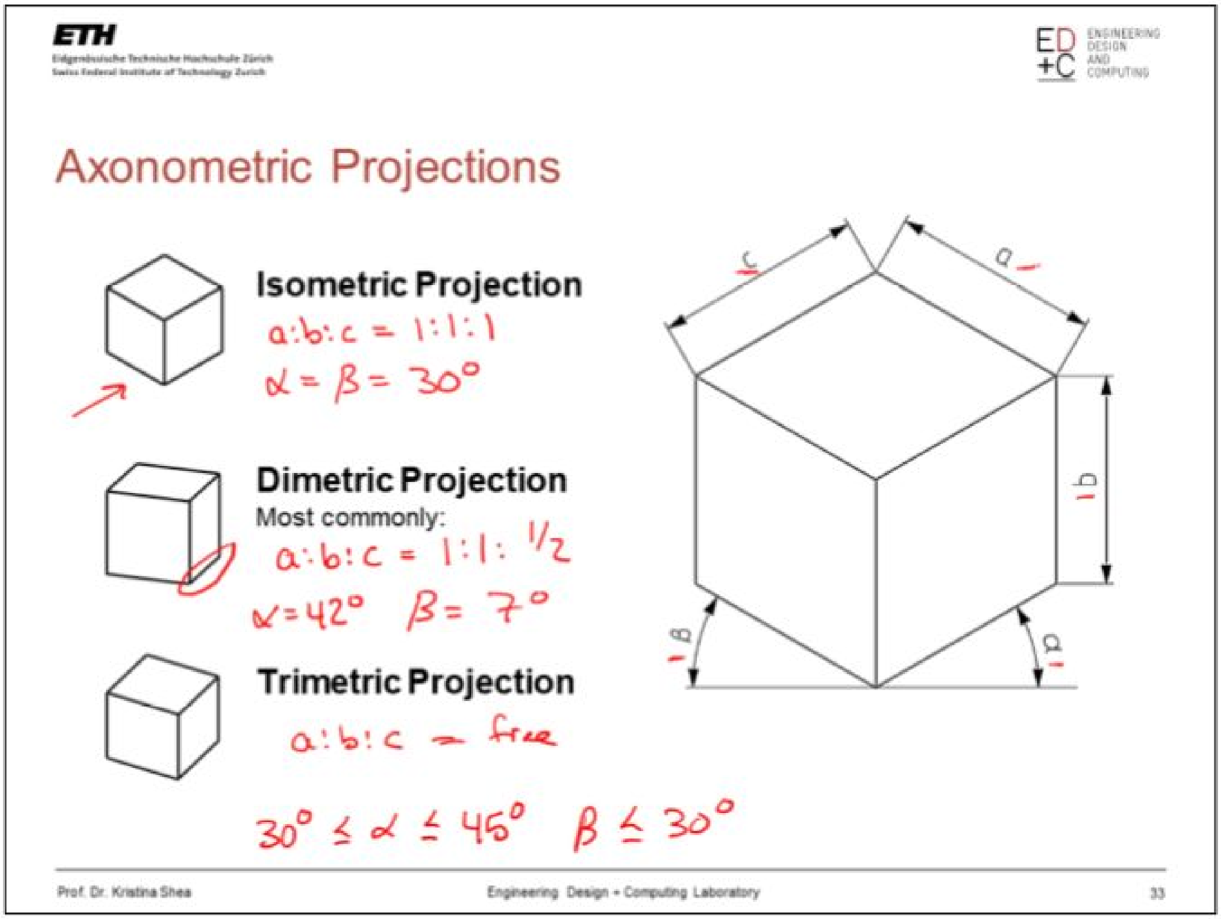



For technical drawings the orthogonal projection (*Orthogonalprojektion*) is the most common.

Here three orthogonal projections in different orthogonal planes (front, top and side) are used to represent the 3D object.

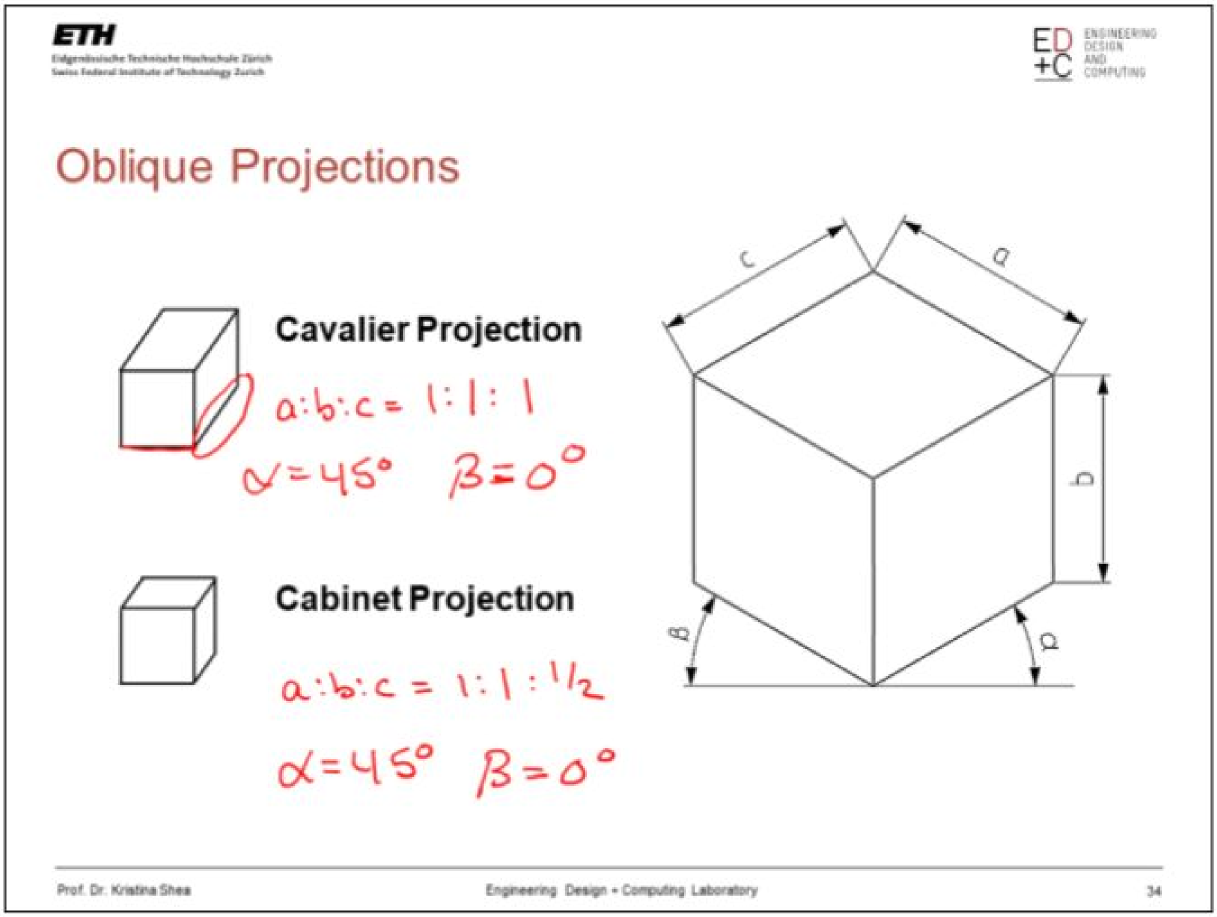
The dimensions of the original 3D object are preserved in the projection.

**Axonometrische Projektion**




**Axonometric projections (*Axonometrische Projektionen*)** are orthogonal projections which try to show all three sides of a 3D object. Depending on the **foreshortening ratios (*Verkürzungsverhältnis*)** (ratio of projected over original segment), we distinguish isometric (*Isometrische*), dimetric (*Dimetrische*) and trimetric (*Trimetrische*) projection. The projection plane intersects the principal axis, therefore, showing more than one face of the object at the same time. In axonometric projections the parallelism of the 3D object lines is preserved but the angles are not. The distances of the object may be represented scaled along different principal axis.

**Schiefwinklige Projektion**




**Oblique projection (*Schiefwinklige Projektion*)** – projectors are not perpendicular to the projection plane (cutting angle equals 45°).

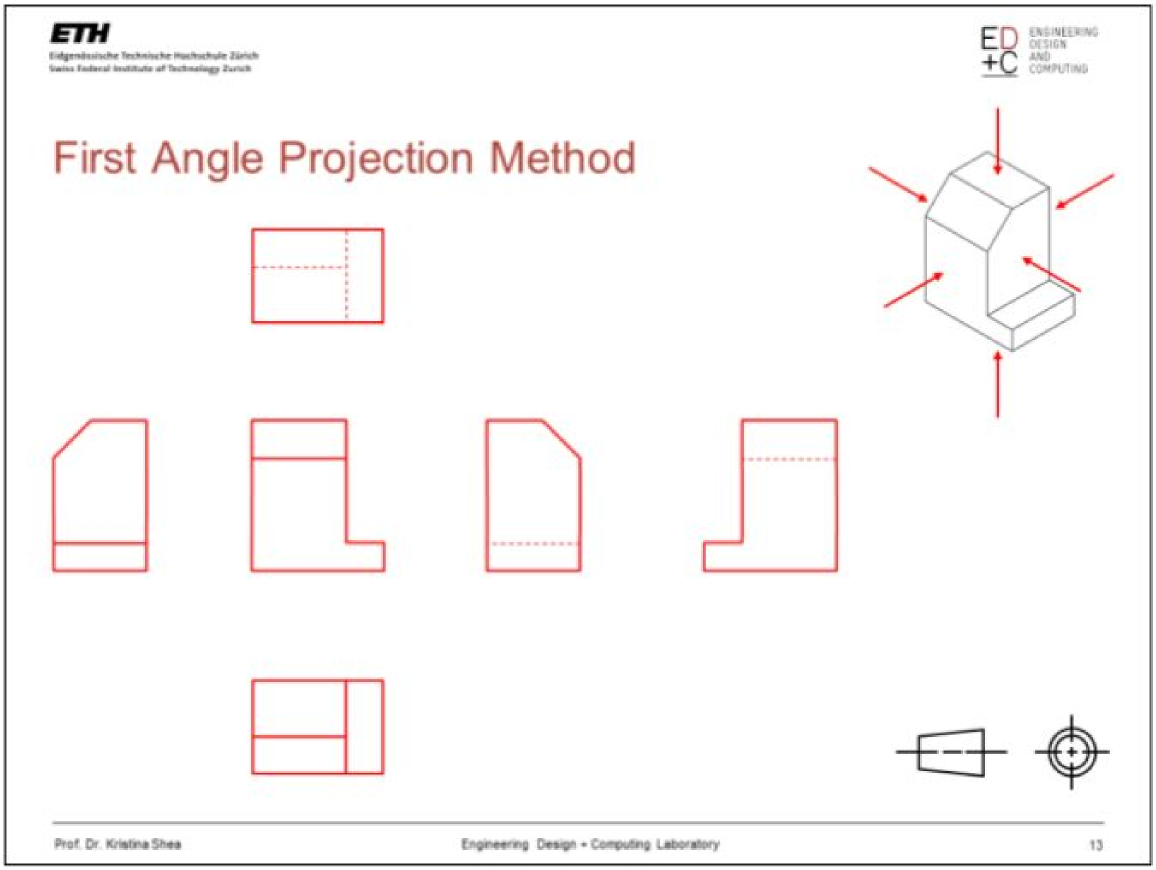
**Cavalier projection (*Kavalier-Projektion*)** - since the foreshortening ratio is set to 1, an object with planar faces perpendicular to, or parallel to, the projection plane appears larger than in reality.

**Cabinet projection (*Kabinett-Projektion*)** – the foreshortening ratio for faces perpendicular to the projection plane is set to ½.

## Projektionsmethoden

**Zusätzliche Informationenen: Technisches Taschenbuch, Schaeffler, Aufl. 2017, Seite 325 ff.**

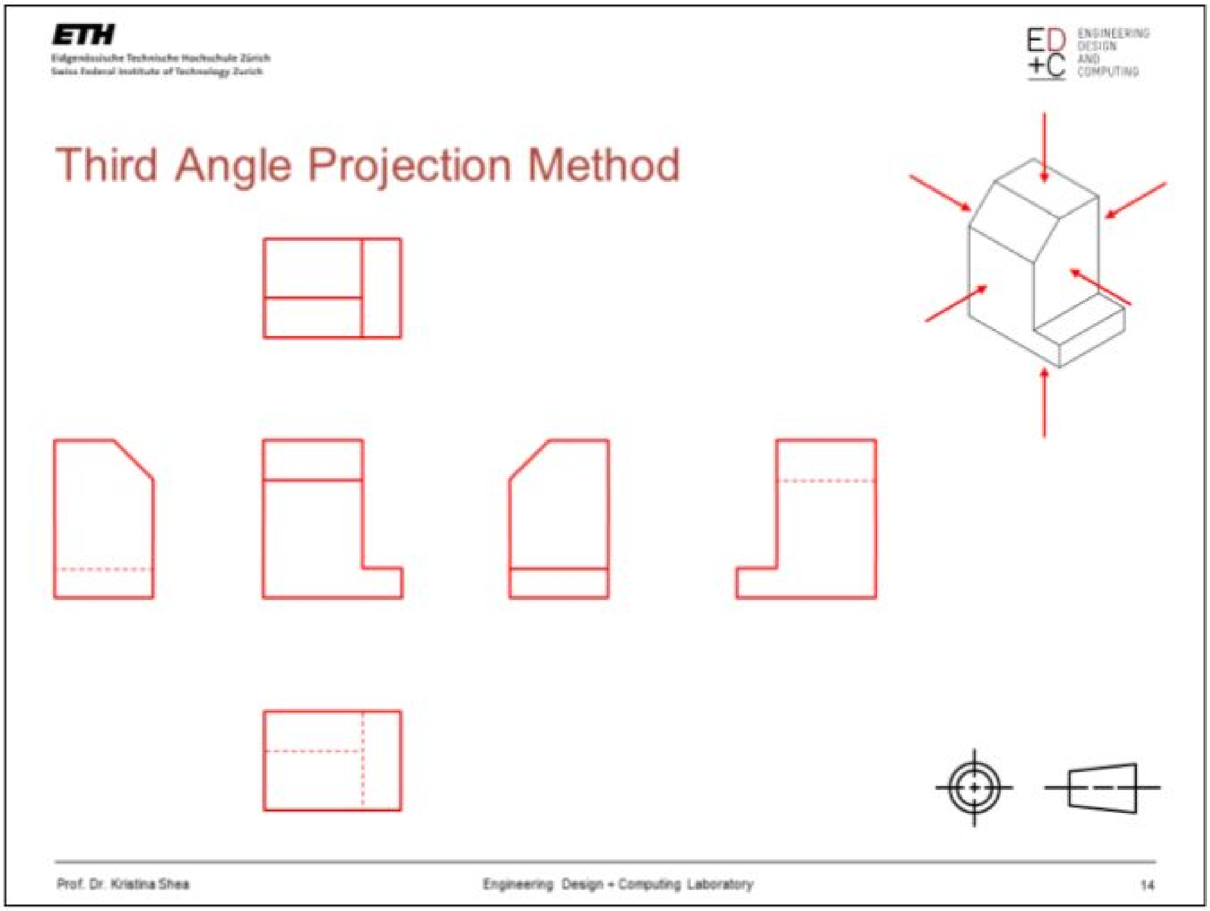
### First Angle Projection Method




Using the first angle projection method (*Projektionsmethode 1*), the arrangement of the orthogonal projections is defined as follows. First, the most informative view is chosen as the principal view. Then, the view from the right is placed to the left and the view from the left is placed to the right of the principal view. The view from the rear is either placed to the far left or right, as convenient. Finally, the view from above is placed underneath and the view from below is placed above the principal view. [SNV Normen-Auszug 2010. Swissmem, Zürich]

The first angle projection method is the standard in Europe (it is also called "European method") and, therefore, used throughout this lecture.

**Third Angle Projection Method**

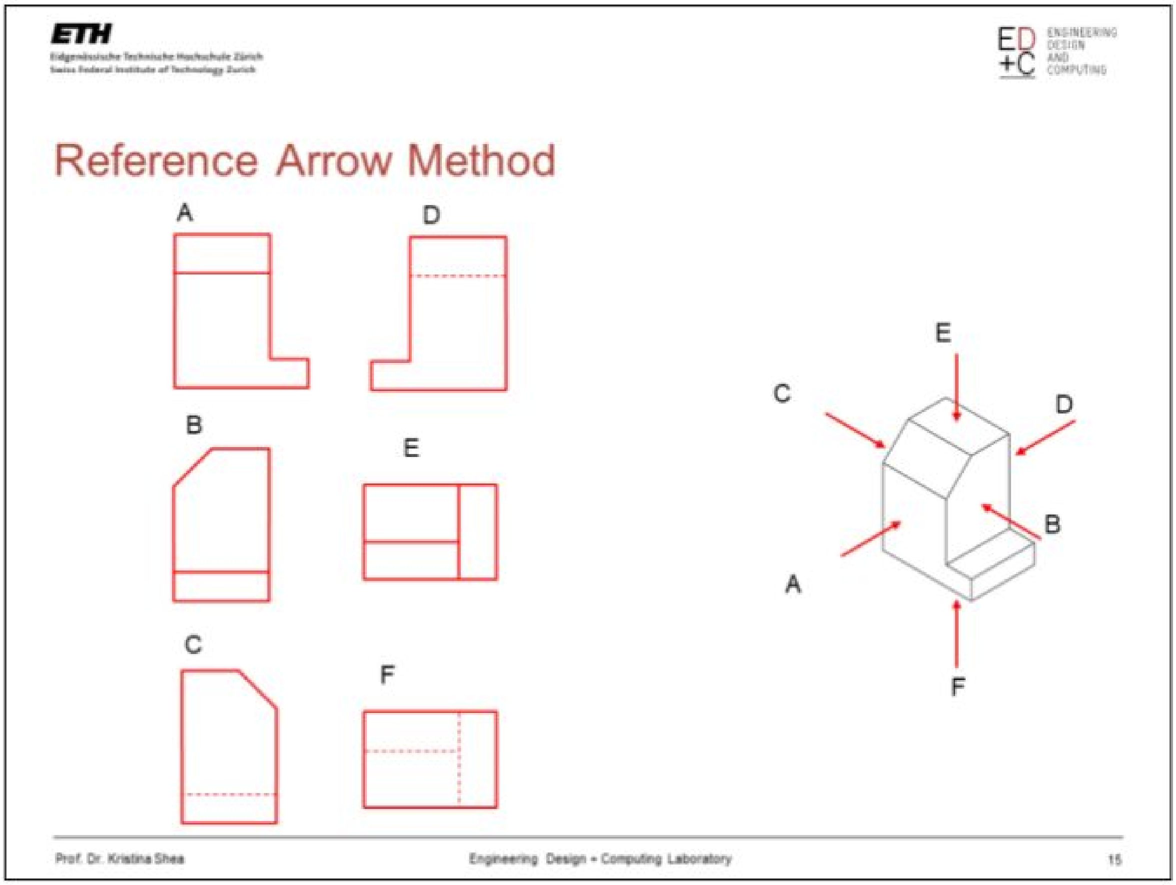



Using the third angle projection method (*Projektionsmethode 3*), the arrangement of the orthogonal projections is defined as follows. First the most informative view is chosen as principal view. Then, the view from the right is placed to the right and the view from the left is placed to the left of the principal view. The view from the rear is either placed to the far left or right, as convenient. Finally, the view from above is placed above and the view from below is placed underneath the principal view.

The reference arrow method, first angle projection method and third angle projection method are the three possible projection methods defined in the ISO 5446-2. The first angle projection method is also called European Method and the third angle projection method US Method.

[SNV Normen-Auszug 2010. Swissmem, Zürich]

**Reference Arrow Method**




Using the reference arrow method, each view is indicated with a reference arrow and identified by a capital letter. The views can be arranged arbitrarily on the drawing. The capital letter must always be printed upright.

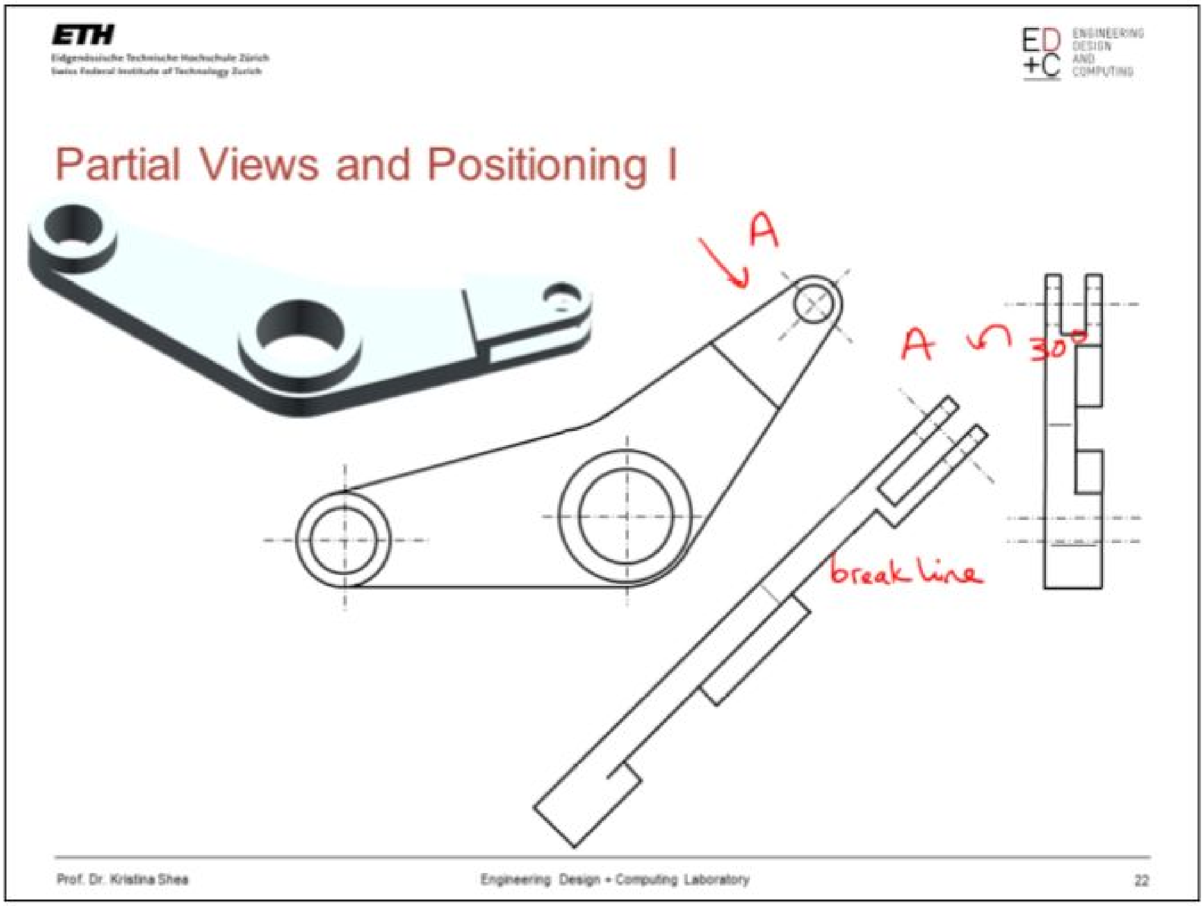
- The outlines and visible edges of the part are illustrated with wide continuous lines (*Volllinie breit*)
- Hidden edges are illustrated with small dashed lines (*Strichlinie schmal*)

[SNV Normen-Auszug 2010. Swissmem, Zürich]

**Arranging Projections**

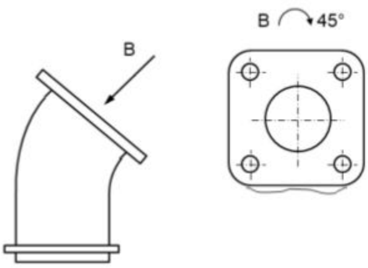
Projections are placed such that they are related to each other.
The spacing between projections is important to allow room for adding dimensions.

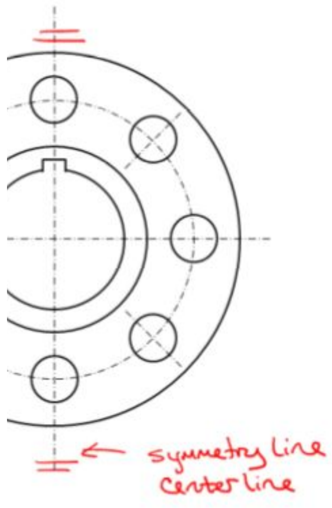
**Teilansichten und Darstellung**




Partial views (*Teilansichten*) allow to avoid repetition in the drawing. Especially, when certain features require a specific illustration but do not need a full view or cannot be represented in one of the existing views, these features may be represented using a partial view. This partial view is limited by a narrow free-hand line (*Freihandlinie schmal*) in drawings done by hand or a continuous narrow line with zigzags (*Zickzacklinie schmal*) in drawings done using CAD tools. [SNV Normen-Auszug 2010. Swissmem, Zürich]

The partial view (*Teilansicht*) is identified with a reference arrow and a capital letter. The size of this capital letter is to be larger than the normal lettering on the drawing by the factor of the square root of two. In order to improve clarity, the partial view may be drawn rotated. This has to be made clear by an arc arrow in the direction of rotation and the angle of rotation. The following indication is used: view identification – arc arrow – angle of rotation (see the example in the slide). If it is not rotated, the partial view has to be drawn in the correct projection as indicated by the reference arrow. [SNV Normen-Auszug 2010. Swissmem, Zürich]

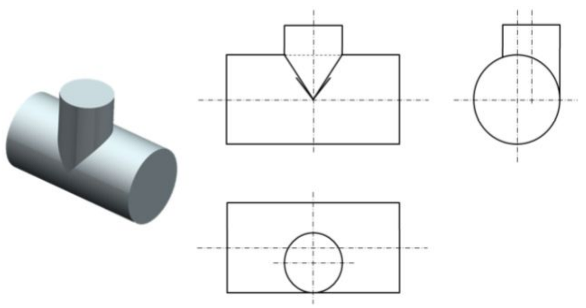





**Symmetrical Views**

In drawings, centerlines (Mittellinien) are represented by narrow long-dash dotted lines (Strich-PunktLinie schmal). The centerline of symmetrical parts is called the symmetry line (Symmetrielinie). Drawings of symmetrical parts can be simplified by only representing one half of the part if the full view is symmetrical. If only one half is drawn, this has to be indicated by adding two narrow short parallel lines orthographic to and at both ends of the symmetry line. [SNV Normen-Auszug 2010. Swissmem, Zürich]

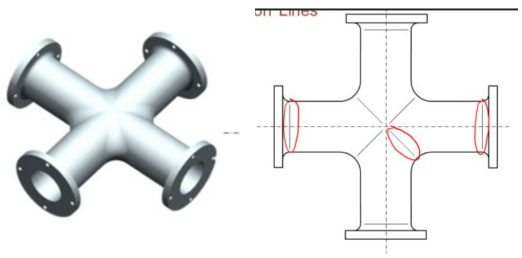
Long-dash dotted lines (Strich-Punkt-Linie schmal) are also used to draw bolt circles.



**Penetrations**

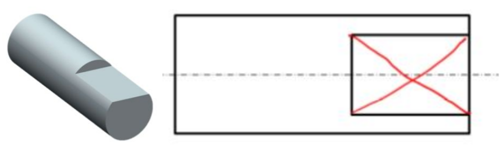
The following slides show special views according to ISO 128-34.

If geometric penetrations (Durchdringung) occur in a part, visible lines of intersection are drawn using continuous wide lines (Volllinie breit) and hidden lines with dashed narrow lines (Strichlinie schmal). To simplify the drawing, the curved lines of penetration can be replaced by straight ones if this simplification does not affect the comprehensibility of the drawing.



**Imaginary Intersections Lines**

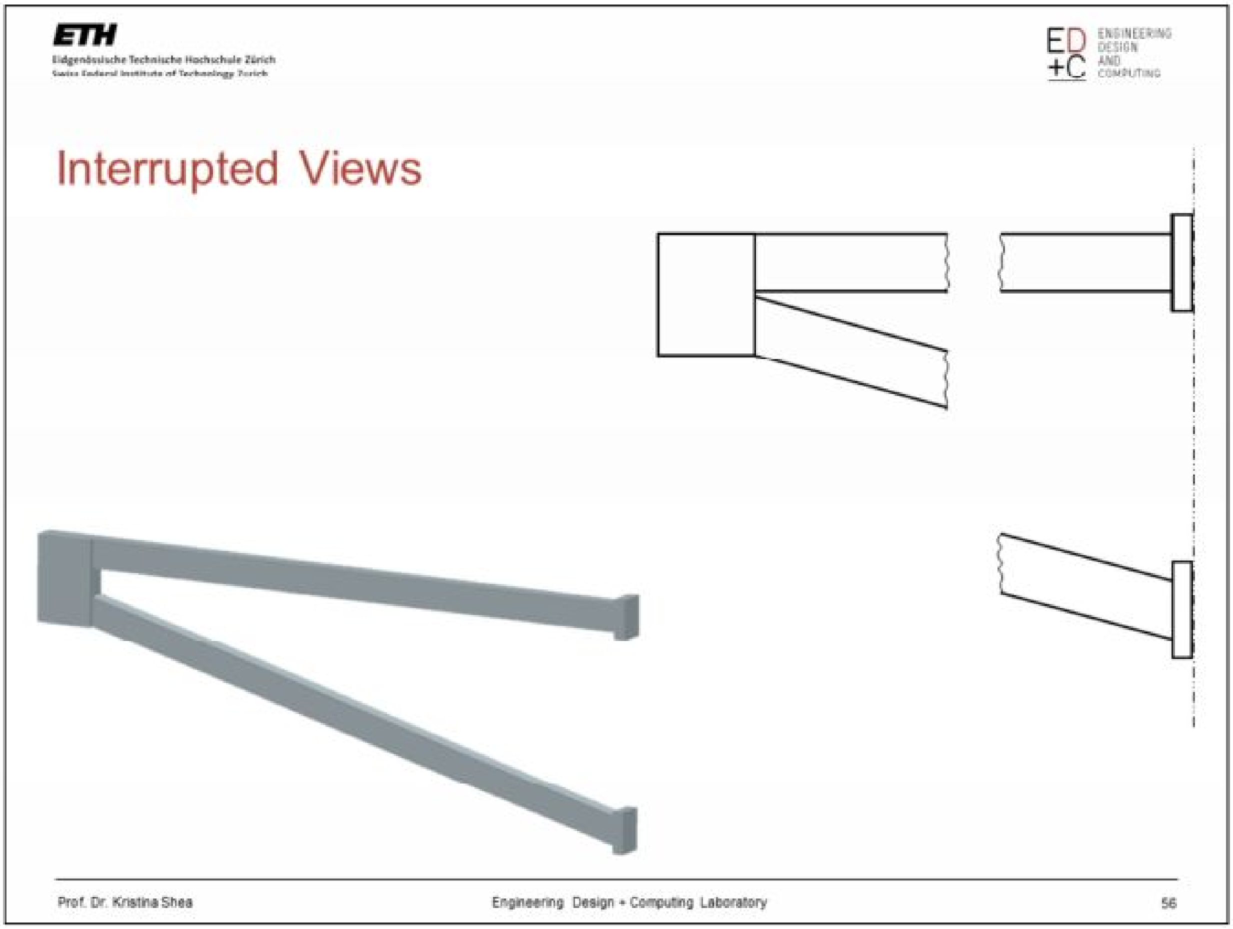
To represent imaginary intersection lines, such as contour edges (Lichtkante) or rounded corners (gerundete Kanten), continuous narrow lines (Volllinie schmal) that do not touch the outlines of the object are used. [SNV Normen-Auszug 2010. Swissmem, Zürich]



**Flat Surfaces on Shafts**

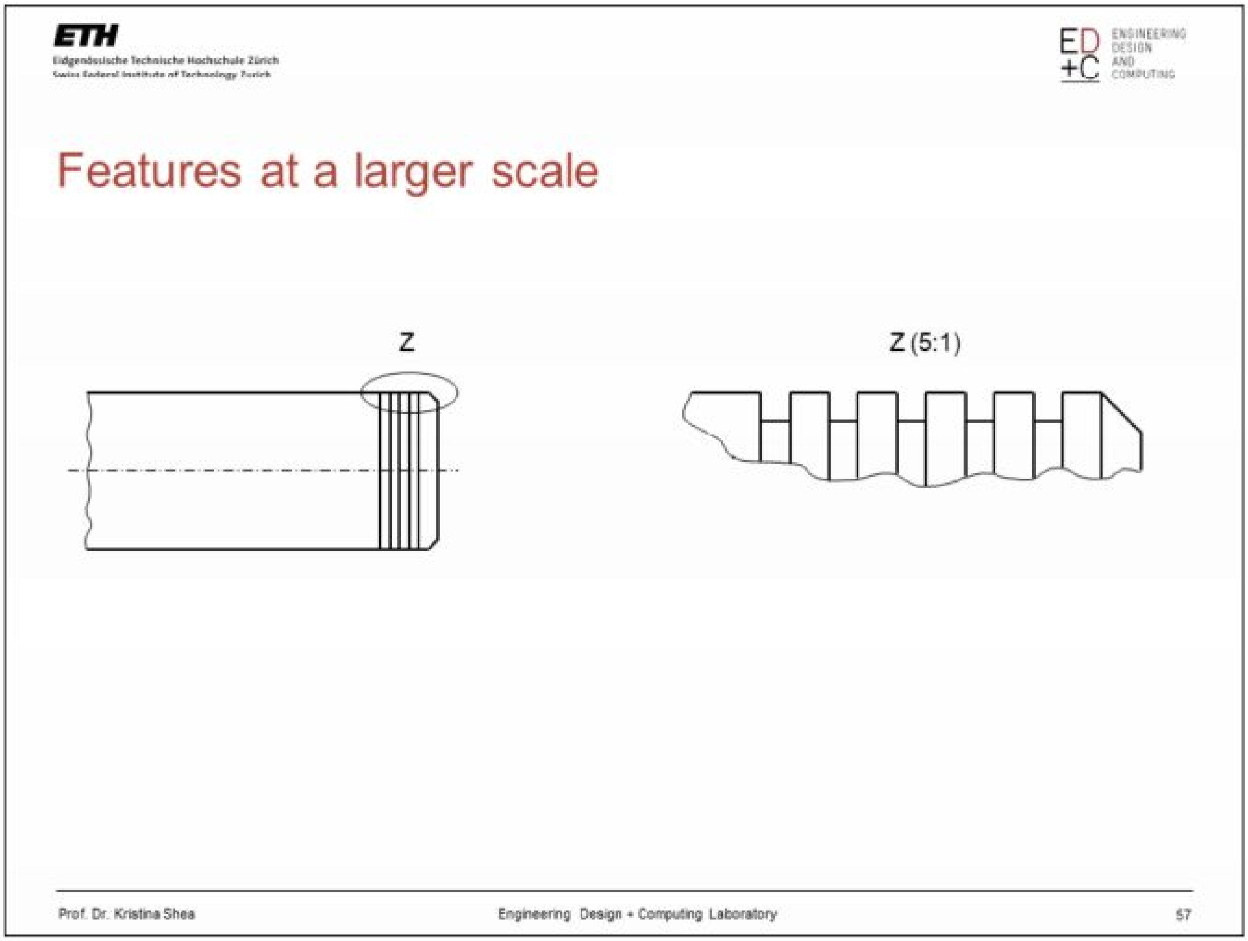
Square ends or flats on shafts can be drawn as diagonal continuous narrow lines (Volllinie schmal). Using this representation, supplementary views, cuts or sections can be avoided.

**Surface Patterns** that appear with knurlings (Rändel), corrugations (Riffel), flutings (Riffelung), meshes (Masche) or lattices (Gitter) have to be drawn using continuous wide lines (Volllinie breit) either completely or partly.



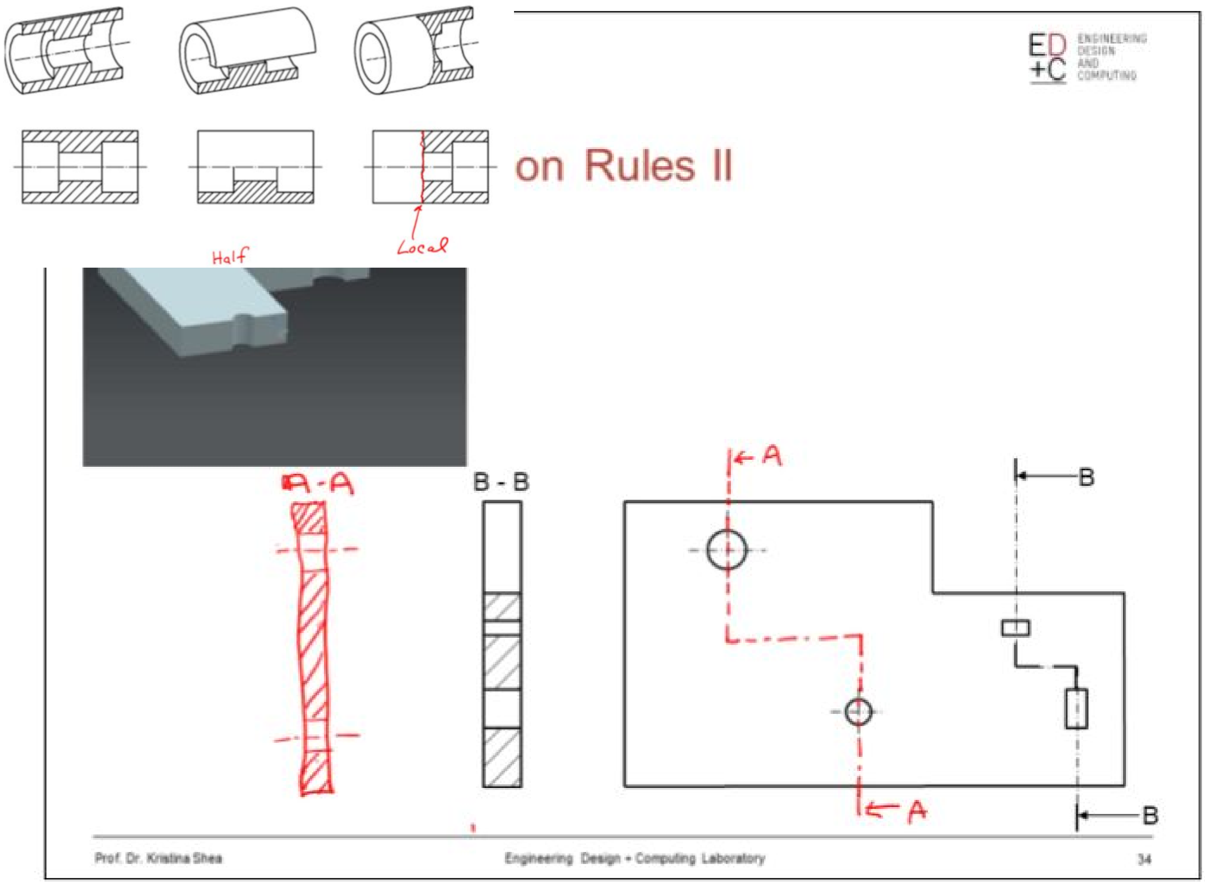

The representation of long objects may be shortened to those parts that are needed for the definition. The limits of the drawn portions are to be drawn as narrow freehand lines (*Freihandlinie schmal*) in drawings done by hand or as zigzag continuous lines (*Zickzacklinie schmal*) in drawings done using CAD tools. The parts are to be drawn close to each other. [SNV Normen-Auszug 2010. Swissmem, Zürich]

Adjacent parts and contours, here a wall on the right side of the part, are illustrated using wide long-dashed double-dotted lines (*Strich-Doppelpunkt-Linie schmal*). [SNV Normen-Auszug 2010. Swissmem, Zürich]




If the scale of the overall drawing does not allow the unambiguous and clear representation of all geometries of an object, these details have to be presented at a larger scale. The areas are encircled with a continuous narrow line (*Volllinie schmal*) and identified with a capital letter. The illustration of the enlarged details can be placed arbitrarily on the drawing but has to be clearly identified with the same capital letter accompanied by the scale of this illustration in parentheses. [SNV Normen-Auszug 2010. Swissmem, Zürich]

## Cuts and Sections




The general rules for the arrangement of views apply equally for sections.

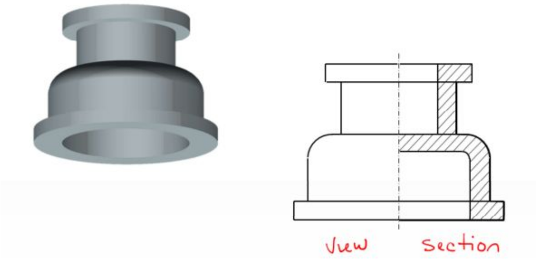
A cutting line (*Schnittlinie*) indicates the position of the cutting plane (*Schnittebene*) or the sequence of two or more cutting planes if existent within the cut. A cutting plane is an imaginary plane that represents the place where the object is cut through. A cutting line is represented by a narrow long-dashed dotted line with wide beginning and ending and continuous wide segments when the direction changes. Straight parts of the cutting line may be omitted if no ambiguity is caused. Each cut and its section is identified by a capital letter that is positioned close to the reference arrows (continuous wide line with 30° tip angle) at the beginning and end of the cutting line. The reference arrows indicate from which side the cutting plane is viewed. The letters have to be oriented such that they are readable in the normal drawing position.

The section (*hier: Schnittfläche*) view has to be drawn in the projection as indicated by the reference arrows. It may be placed arbitrarily on the drawing. Each section has to be identified by the corresponding capital letters separated by a dash. To indicate where solid material is cut, the resulting areas in the section view are hatched (*schraffiert*).

- **Cutting plane** (*Schnittebene*): imaginary plane at which the object is cut
- **Cutting line** (*Schnittlinie*): line indicating the position of one or the sequence of all cutting planes that belong to a cut
- **Section** (*hier: Schnittfläche*): view showing only outlines of an object where it cuts the cutting plane(s). If necessary, contours that lie behind the cutting plane(s) may also drawn.

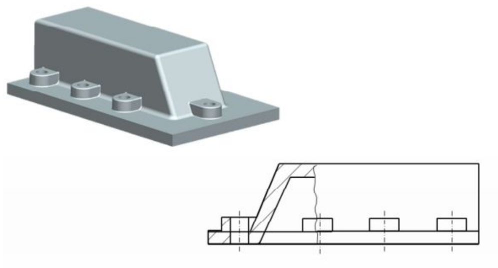
### Types of Sections

There are three main types of sections:
• (Full) Section (Vollschnitt)
• Half Section (Halbschnitt)
• Local Section (Teilschnitt)



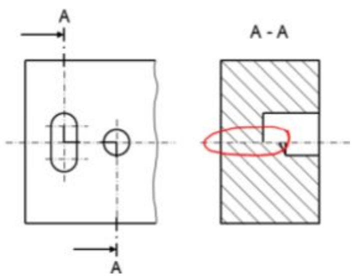

**Half-Section of symmetrical Parts**

If a part is symmetrical, it may be drawn half in view and half in section. In these, so-called, half sections (Halbschnitt) no hidden edges are illustrated and the cutting line is not indicated. The same is valid for sections of rotationally symmetrical parts.

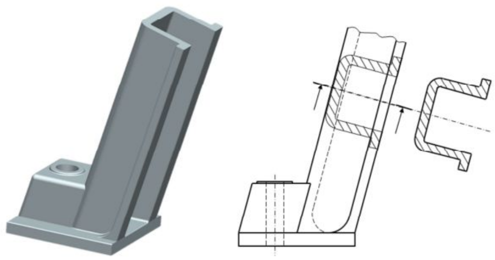


**Local Section**
In cases when only small details need clarification, local sections (Teilschnitt) can be used. Such a local section is limited by a zigzag (in drawings made in CAD) or freehand (in drawings made manually) continuous narrow line.
Local Section (Teilschnitt): representation in which only a part of an object is represented in section.

In sections, the areas of solid material are illustrated using hatching (Schraffur). For hatching continuous narrow lines are used, preferably at a 45° angle to the principal outlines or lines of symmetry.
If several areas of section cut the same part, they all have to be hatched identically. If, by contrast, several parts are cut, adjacent parts are hatched in different directions and/or differently spaced. The space between the hatching lines is preferably chosen in proportion to the size of the hatched area.
For large areas it is allowed to hatch only a suitable zone following the contour of the area.




If a part is cut with two parallel cutting planes, the hatching in the area of section must be identical, but can be offset along the dividing line to achieve greater clarity.

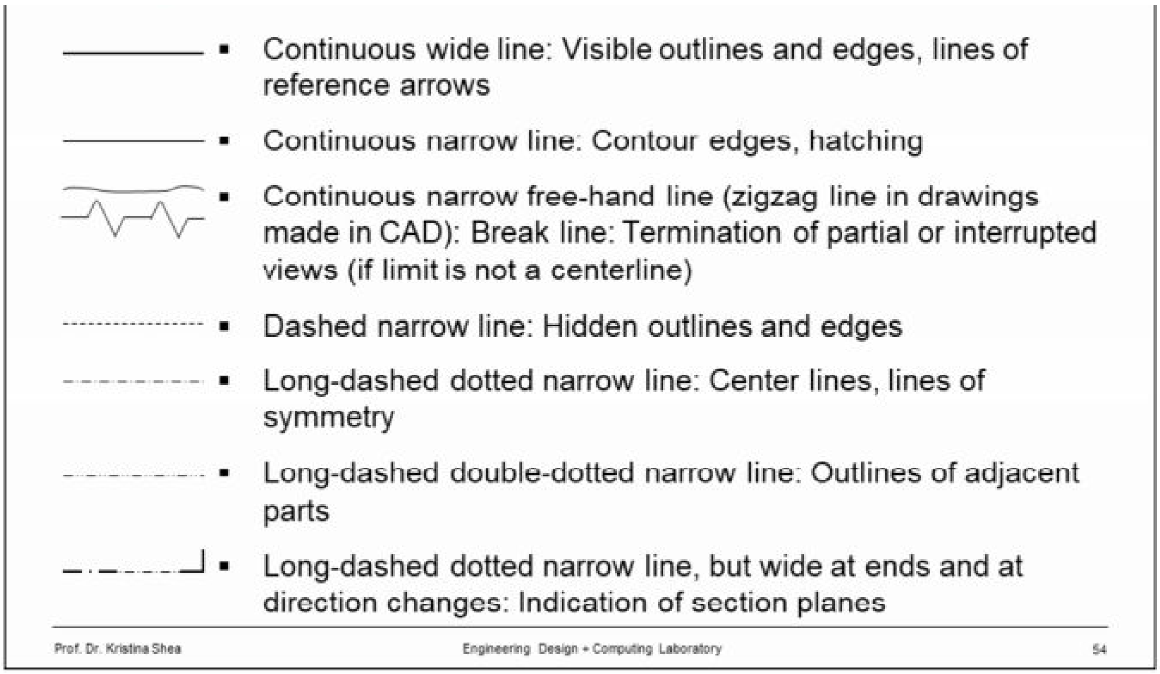


**Rotated Sections within view**

A section may be rotated into the corresponding view, if no ambiguity arises. In this special case, the section's outlines are to be drawn in continuous narrow lines, the section needs no identification and no indication of the direction of viewing.

Rotated sections can be detached from a view with no identification as long as they are placed near that view and connected to it by a long-dashed dotted narrow line.

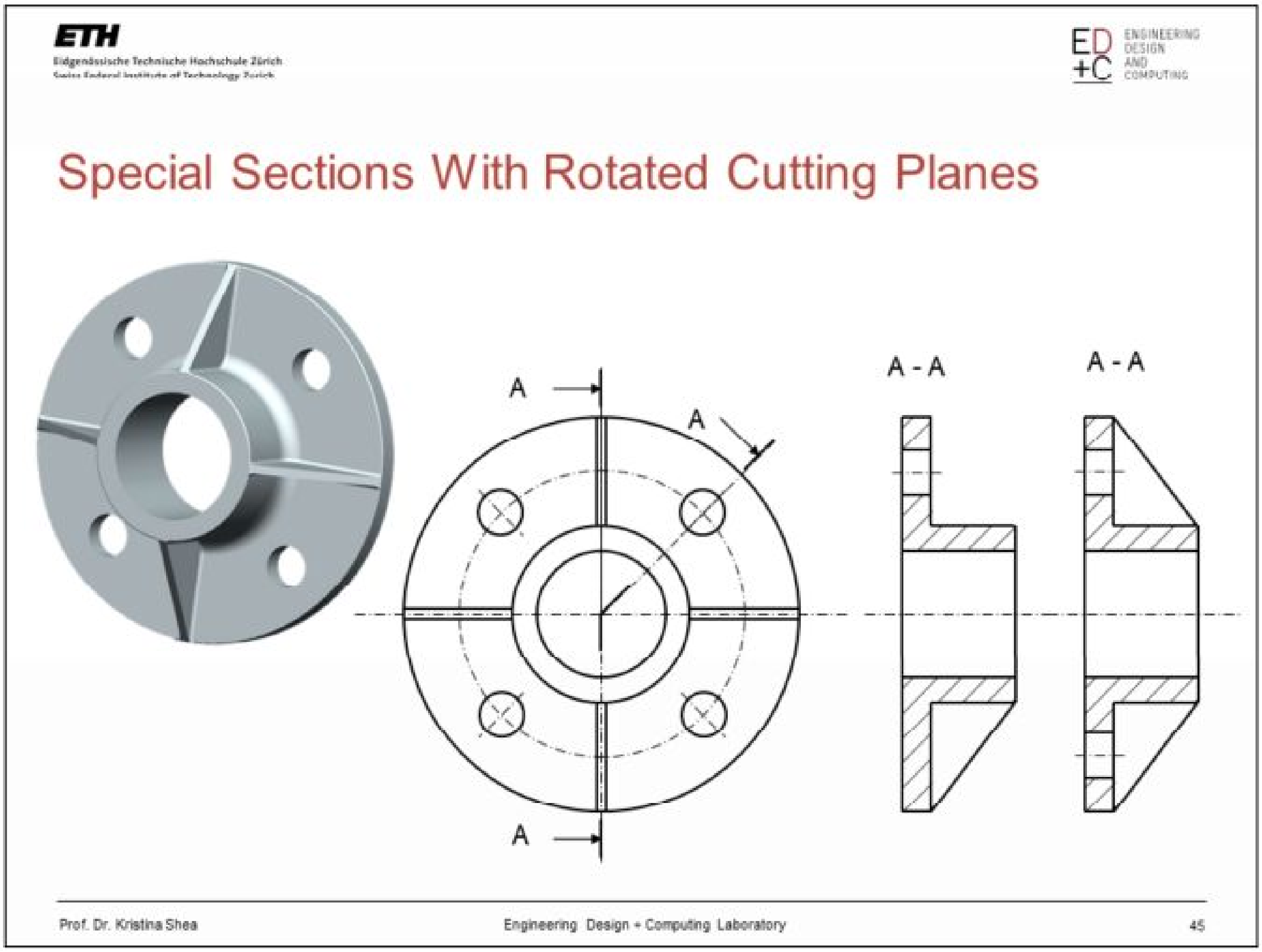
**Line Types**




All definitions according to [SNV Normen-Auszug 2010. Swissmem, Zürich]

- Continuous wide line (*Volllinie breit*)
- Continuous narrow line (*Volllinie schmal*)
- Continuous narrow free-hand line (*Freihandlinie schmal*)
- Continuous narrow zigzag line (*Zickzacklinie schmal*)
- Dashed narrow line (*Strichlinie schmal*)
- Long-dashed dotted narrow line (*Strichpunktlinie schmal*)
- Long dashed double-dotted narrow line (*Strich-Zweipunktlinie schmal*)

**Rotated Cuts**




<u>First cut:</u>

The first cut shows a section that is defined by two intersection cutting planes. One of the planes is rotated into the plane of projection.

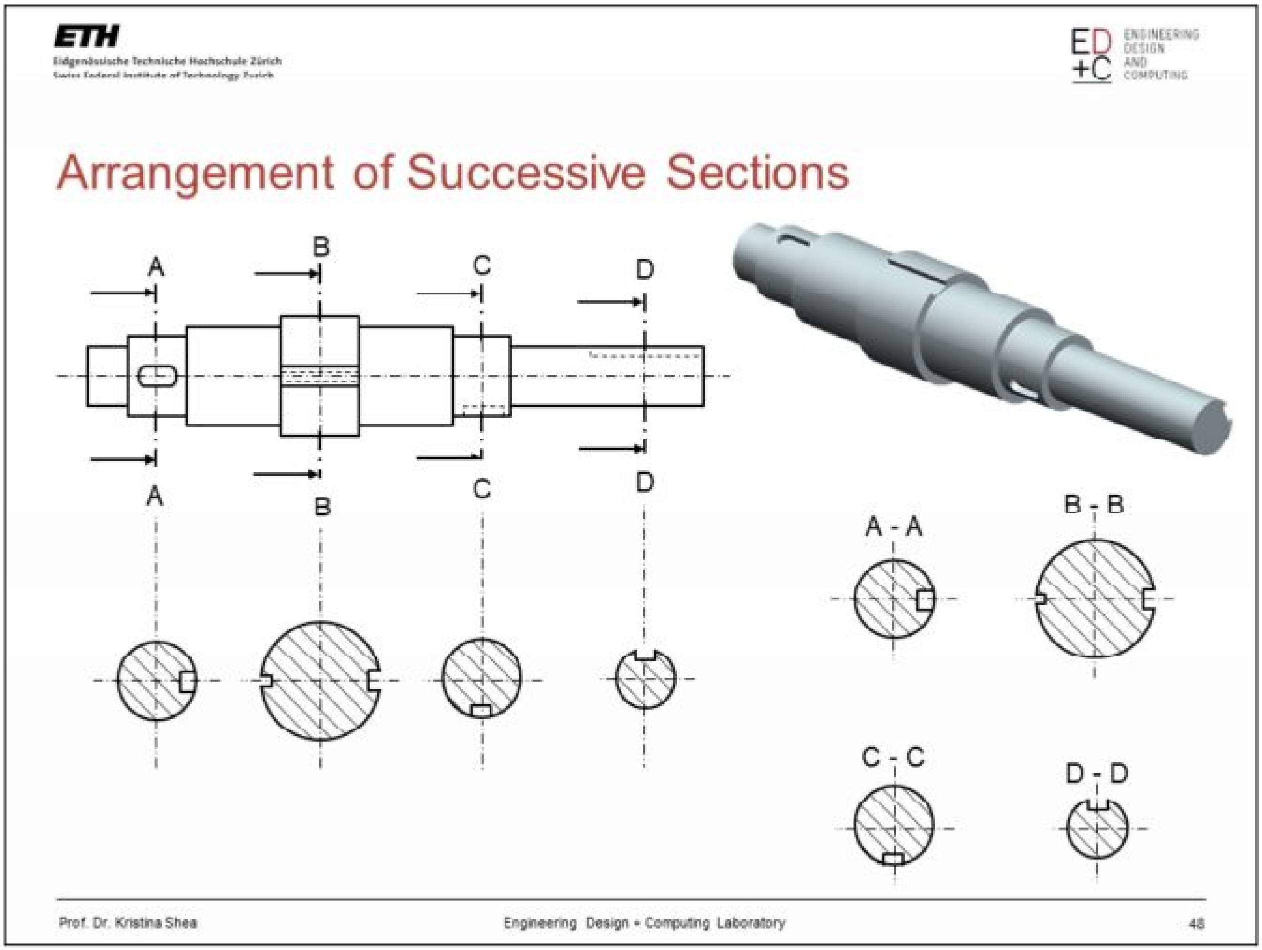
<u>Second cut:</u>

If the part that is to be drawn contains regularly spaced details, such as holes of flanges or bearing covers, etc., these details may be rotated into the cutting plane. This avoids ambiguity since the need for several or more complicated sections is removed.

<u>Additional:</u>

Details such as ribs (*Rippen*), fasteners (*Befestigungselemente*), shafts (*Wellen*), spokes of wheels (*Radspeichen*) and similar are not cut.

**Arrangement of Successive Sections**




The part on the slide is a single part, a stepped shaft that is, e.g., machined on a lathe. The arrangement of successive sections (*aufeinanderfolgende Schnitte*) is presented in the slide above. Either the sections are identified and may be positioned arbitrarily on the technical drawing, or a succession of removed sections can be used.

Outlines and edges that lie behind the cutting plane may not be represented if they are not necessary for the clarity of the drawing.

## Dimensions

**...for function**

- Dimensions required to fulfill specific functions
- Dimensions necessary for interconnections between parts
- Tolerances define correct fits (Lecture 10)

**…for manufacturing**

- Dimensions depend on manufacturing method
- Dimensions are to be given such that they can be used directly for manufacturing the part without any calculations or unit conversions, e.g. choosing a workpiece (raw material size) from the maximum dimensions on the drawing.
- Most drawings of single parts are drawings for manufacturing.

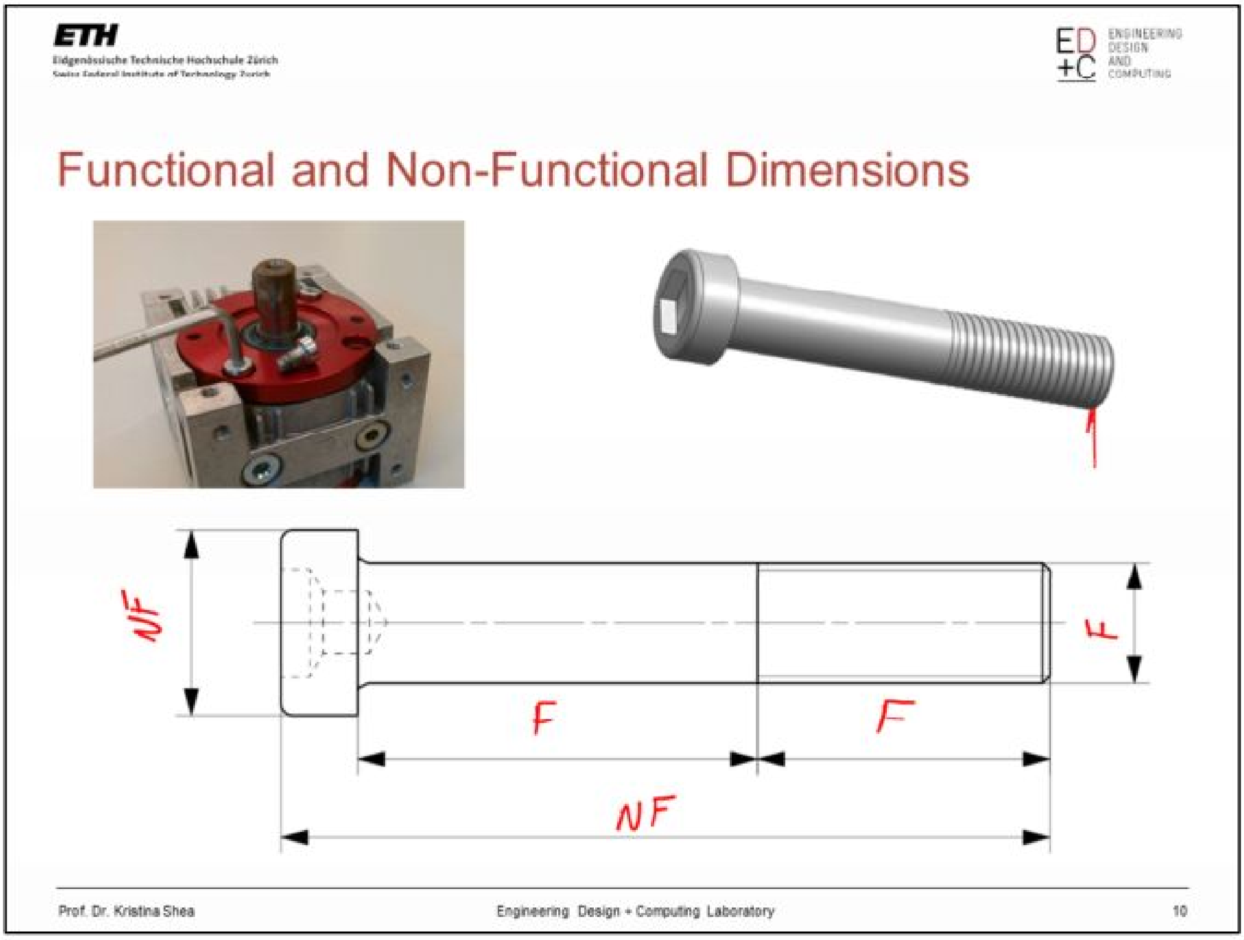
**...for inspection**

- Necessary to verify dimensions
- Depends on the measuring instruments used
- Dimensions are to be usable without calculations or unit conversions

**General rules for dimensioning**

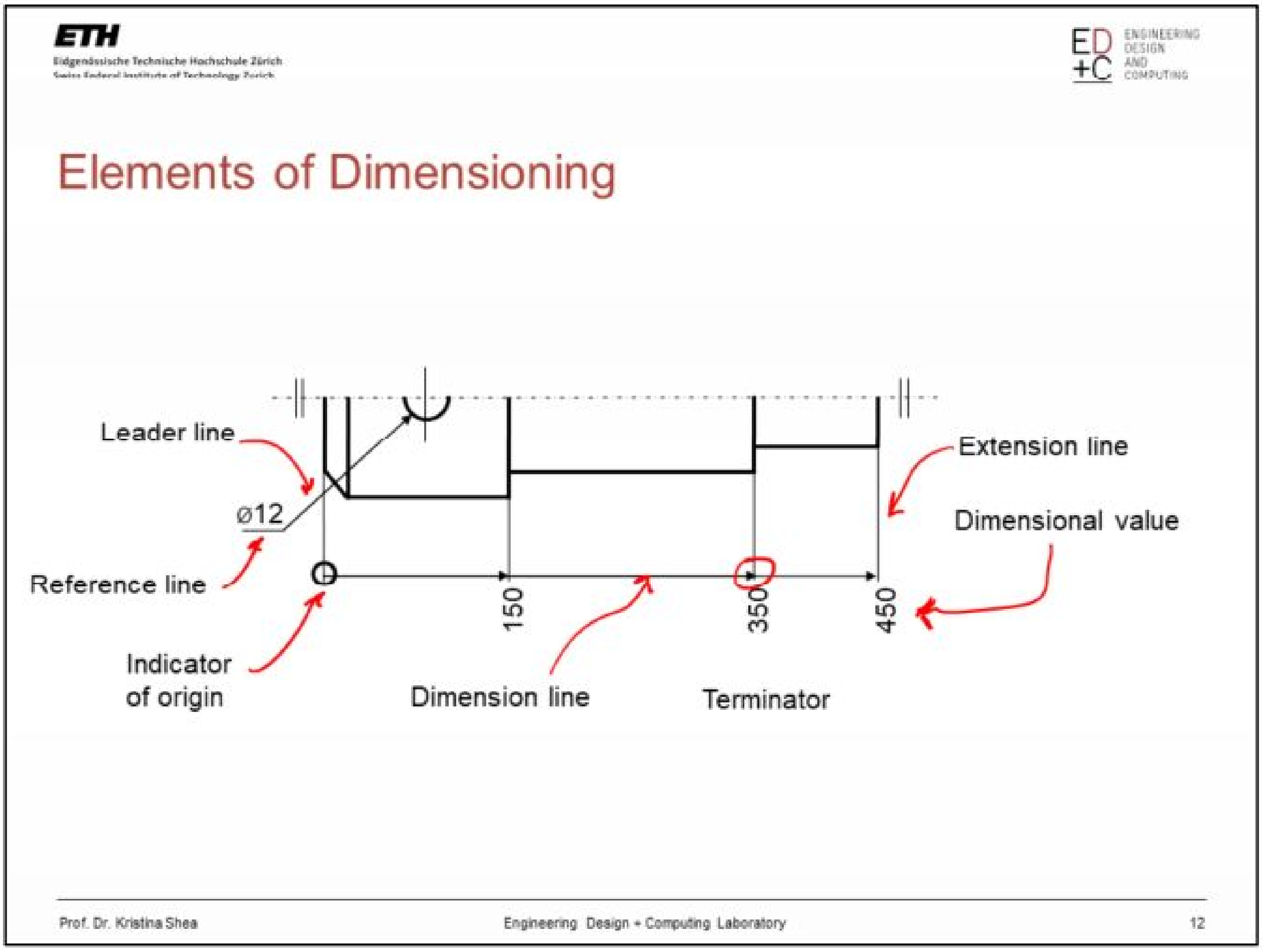
- Dimensions should be given in the view or section where the particular element is best represented
- Every dimension must be given only once  In a technical drawing, the represented part has to be dimensioned completely, i.e. unambiguously
- Dimensions are given in mm by default  Function, manufacturing and inspection have to be considered for dimensioning

**Functional and Non-Functional Dimensions**




**Functional dimensions (*Funktionsmasse* ) (F):**
- Define **form, size and position** of **features important for specific functions**
- Have to be given in all drawings
- Have to be toleranced (Lecture 10)
- Can be derived from the part interactions to fulfill specific functions

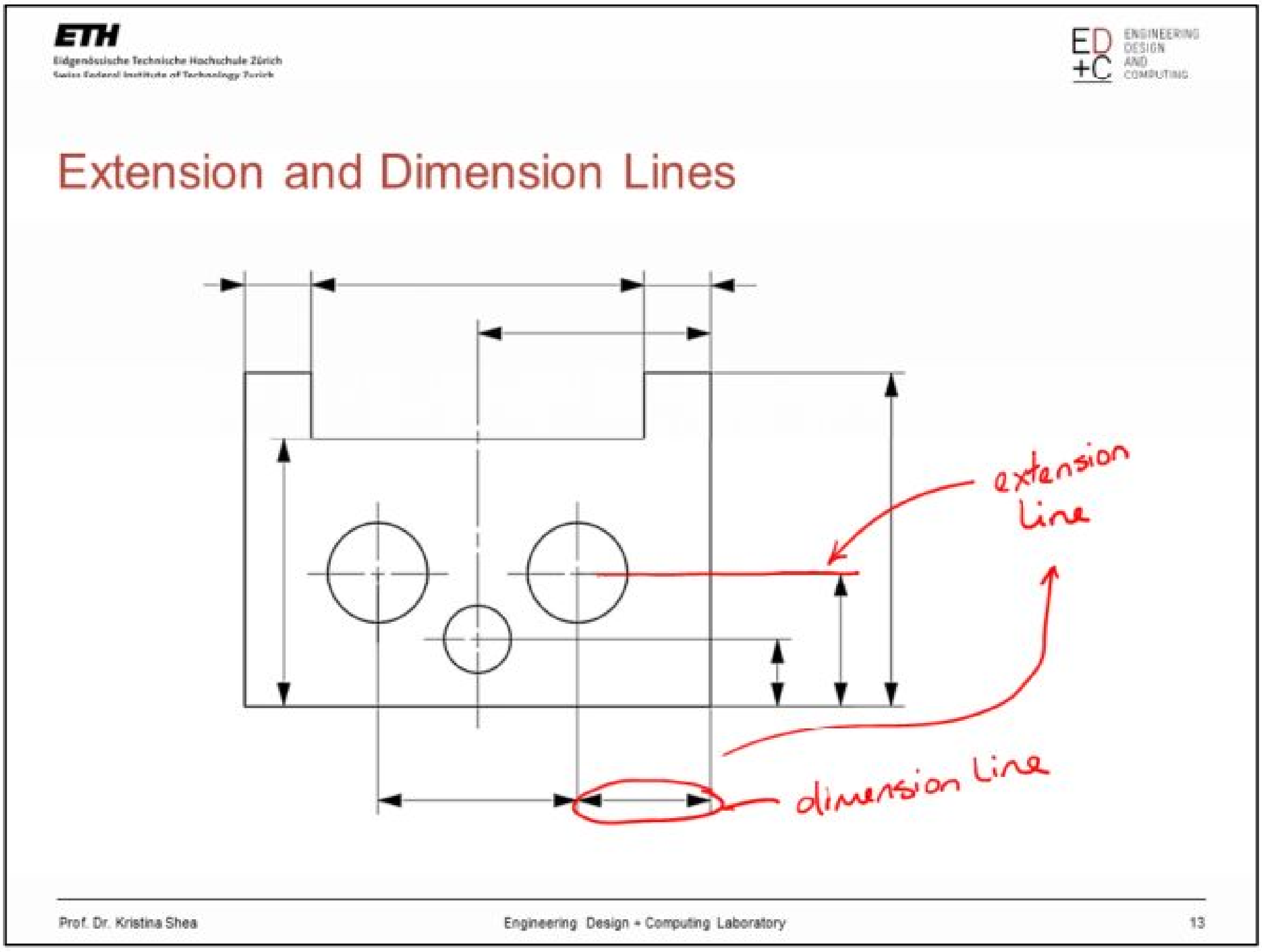
**Non-functional dimensions (*Nichtfunktionsmasse*) (NF):**
- Define **form, size and position of all other features**
- Tolerances do not have to be defined explicitly, but are implicitly defined by the general tolerances of the drawing




These are the elements used for dimensioning. They are further explained on the following slides

- Leader line (*Hinweislinie*)
- Reference line (*Bezugslinie*)
- Indicator of origin (*Ursprung der Bemassung*)
- Dimension line (*Masslinie*)
- Terminator (*Masslinienbegrenzung*)
- Dimensional value (*Masszahl*)
- Extension line (*Masshilfslinie*)

### Extensions and Dimensions Lines




**Extension line (*Masshilfslinie*):** – continuous narrow line
- Perpendicular to the element that has to be dimensioned
- Extended a little (eight times the line thickness)
- Center lines and contour lines can be used as extension lines
- Can be interrupted when their continuation is unambiguous

**Dimension line (*Masslinie*):** – continuous narrow line
- Parallel to the element that has to be dimensioned
- Outside the contours of the part (only inside if clear)
- Should not be intersected with other lines
- Center lines and contour lines can **never** be used as dimension lines
- Should be in 10 mm distance from the contours of the object
- The distance between several dimensions lines in parallel should always be the same and minimum 7 mm

**Line terminators (*Masslinienbegrenzung*) :** – arrow head with 30° head angle
- If there is not enough space for arrows and dimension, the arrows can be placed outside

**Auxiliary dimensions (Hilfsmasse) (H):**

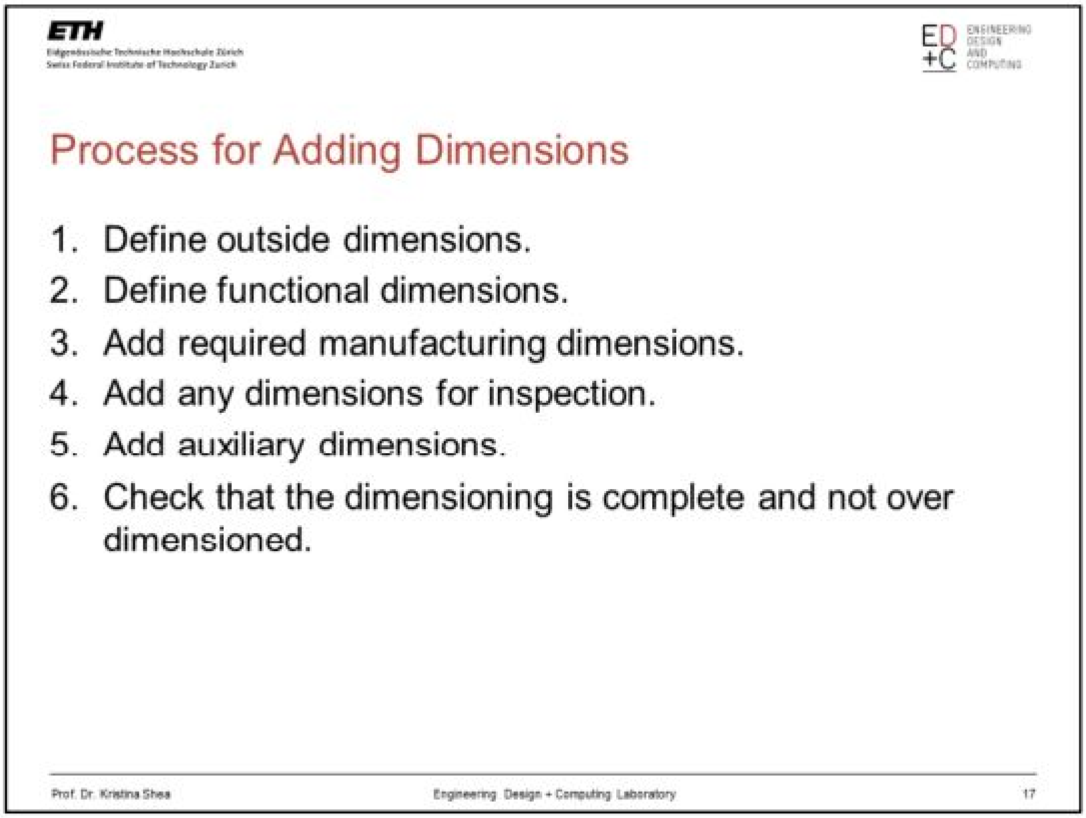
- Are dimensions that give additional information to avoid calculations
- Do not have to be toleranced
- Are given in parentheses

**Dimensional values (Masszahl):**

- same (readable) size throughout the whole drawing
- Above and separated from dimension line, in the middle, not divided or crossed by lines
- Read from the bottom or from the right
- When there is not enough space above the dimension line, give the dimensional value using a leader lin

**Summary for dimensioning:**

- Every geometric element has to be described with dimensions unambiguously
- Main dimensions, functional dimensions and auxiliary dimensions are used
- No over determined dimensioning is allowed (use auxiliary dimensions to avoid this), e.g. in the example on the slide, one of the three horizontal dimensions has to be an auxiliary dimension.

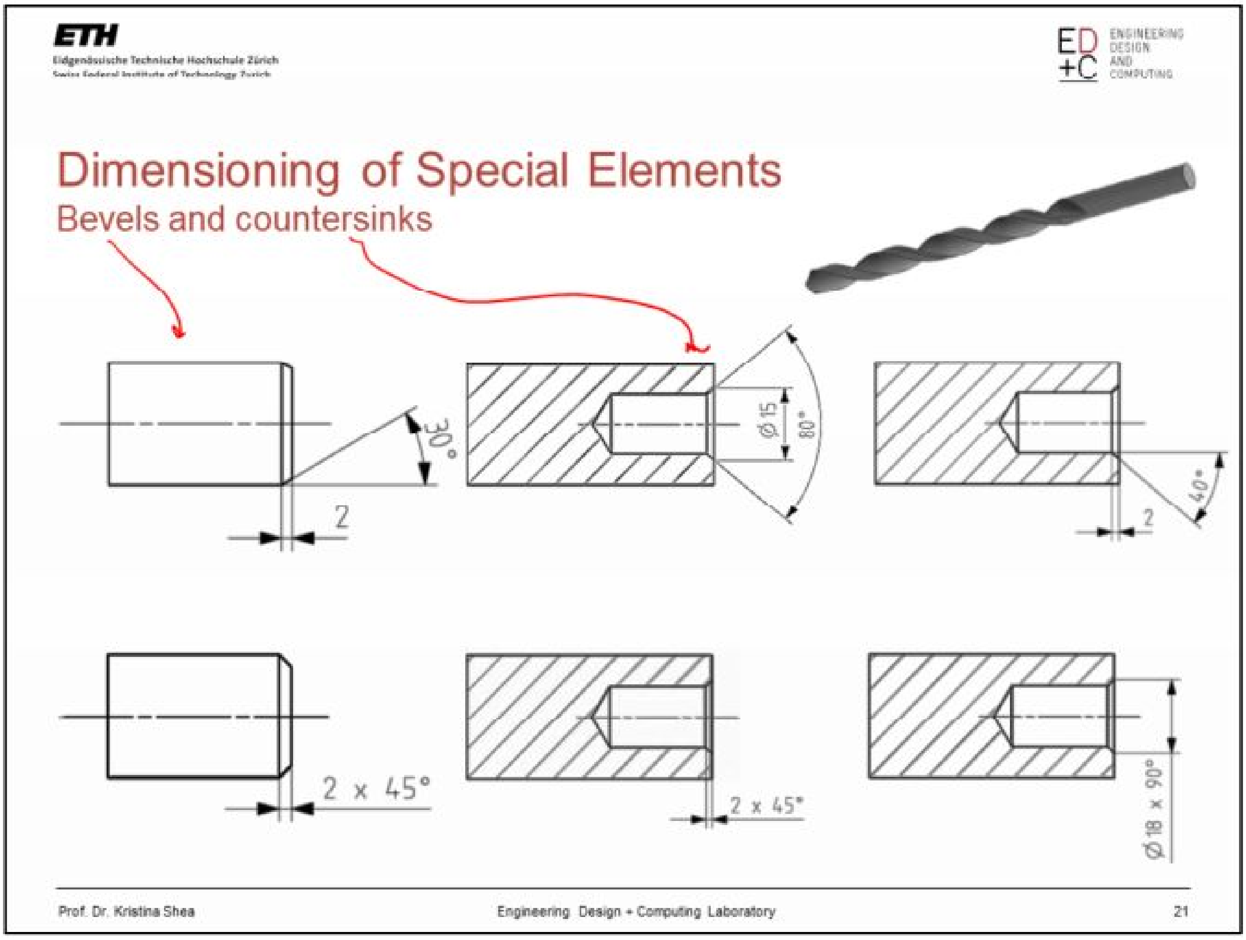


**Arrangement of Dimensions**

- Chain dimensioning (Kettenbemassung)
- Parallel dimensioning (Parallelbemassung)
- Running dimensioning (Stufenbemassung)
- Combined dimensioning (Kombinierte Masseintragung)

**Zusätzliche Informationenen: Technisches Taschenbuch, Schaeffler, Aufl. 2017, Seite 342 f.**

**Dimensioning for Special Elements**

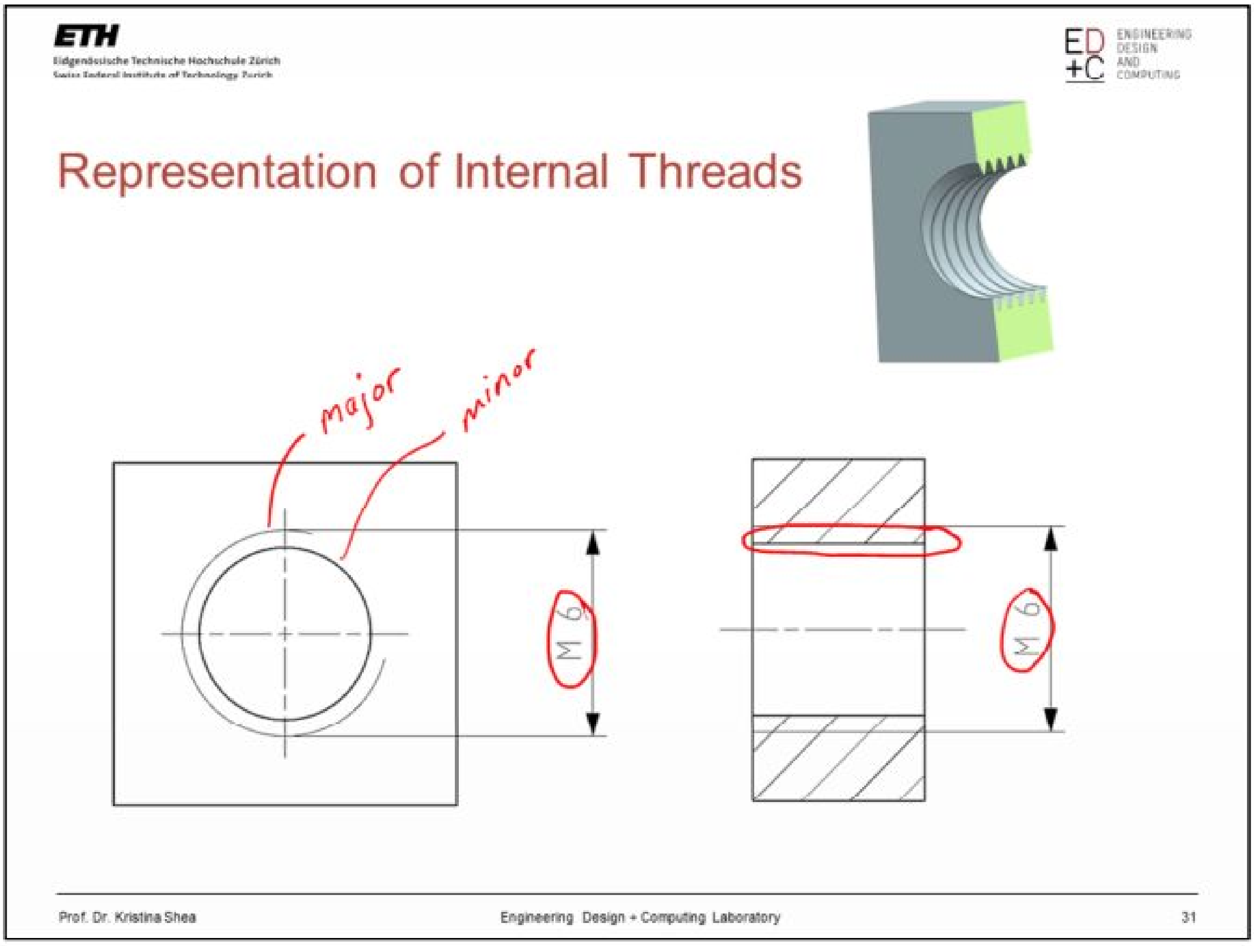



**Special dimensioning:**

- Dimensioning for bevels (*Anschrägungen*) and countersinks (*Ansenkungen*) is shown above

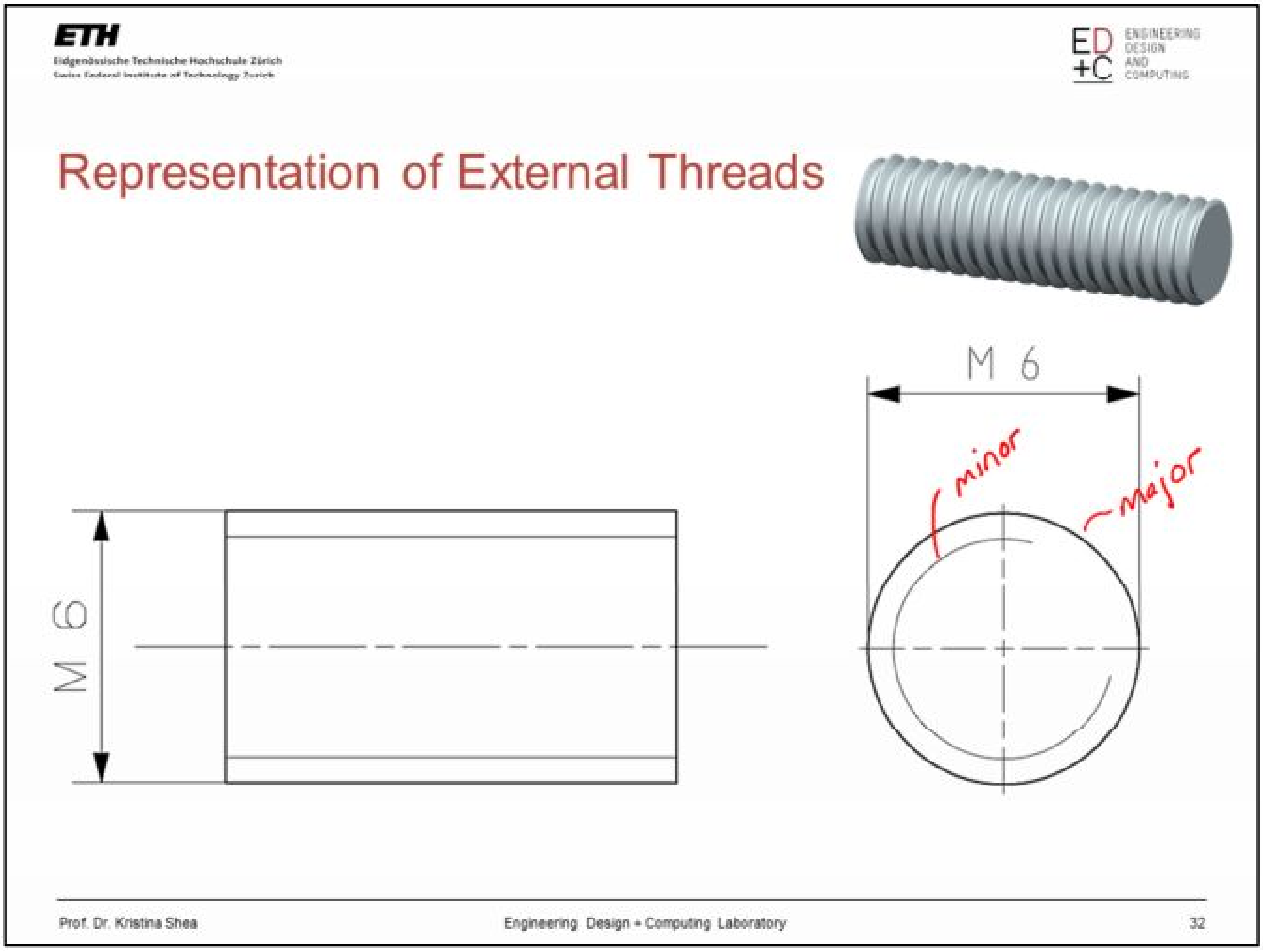
**Simplified dimensioning:** - for 45° and 90° only

- When the angles are 45° or 90°, simplified dimensioning can be used, as shown in the examples at the bottom



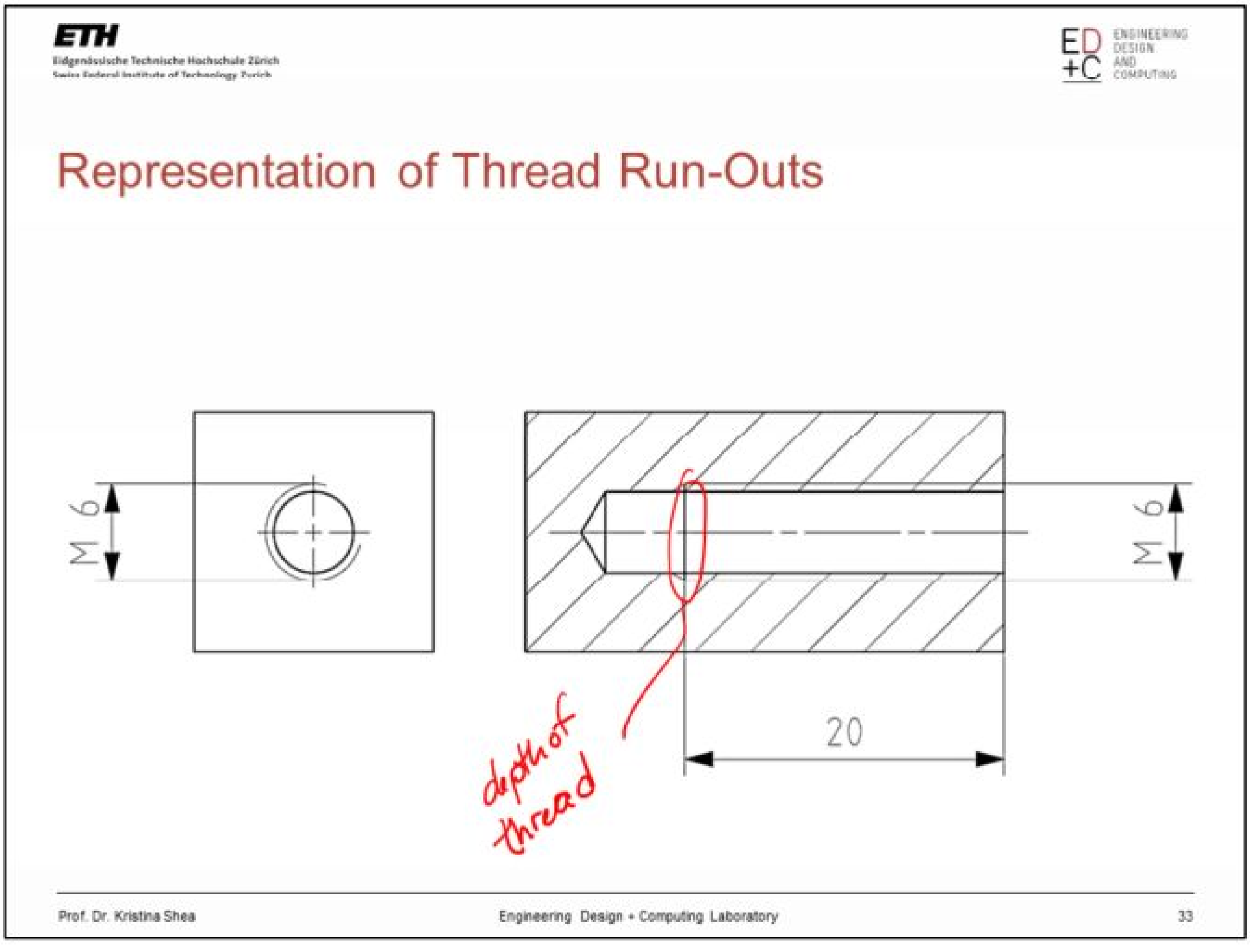

**Internal thread (*Innengewinde*)**:

- The thread is represented using the symbol **M** (for metric threads) preceding the dimensional value
- Major diameter (*Gewindeaussendurchmesser*): continuous **narrow** line
- Minor diameter (*Gewindekerndurchmesser*): continuous **wide** line
- End view: major diameter is shown as a ¾ circle
- Hatching: extended to **minor** diameter



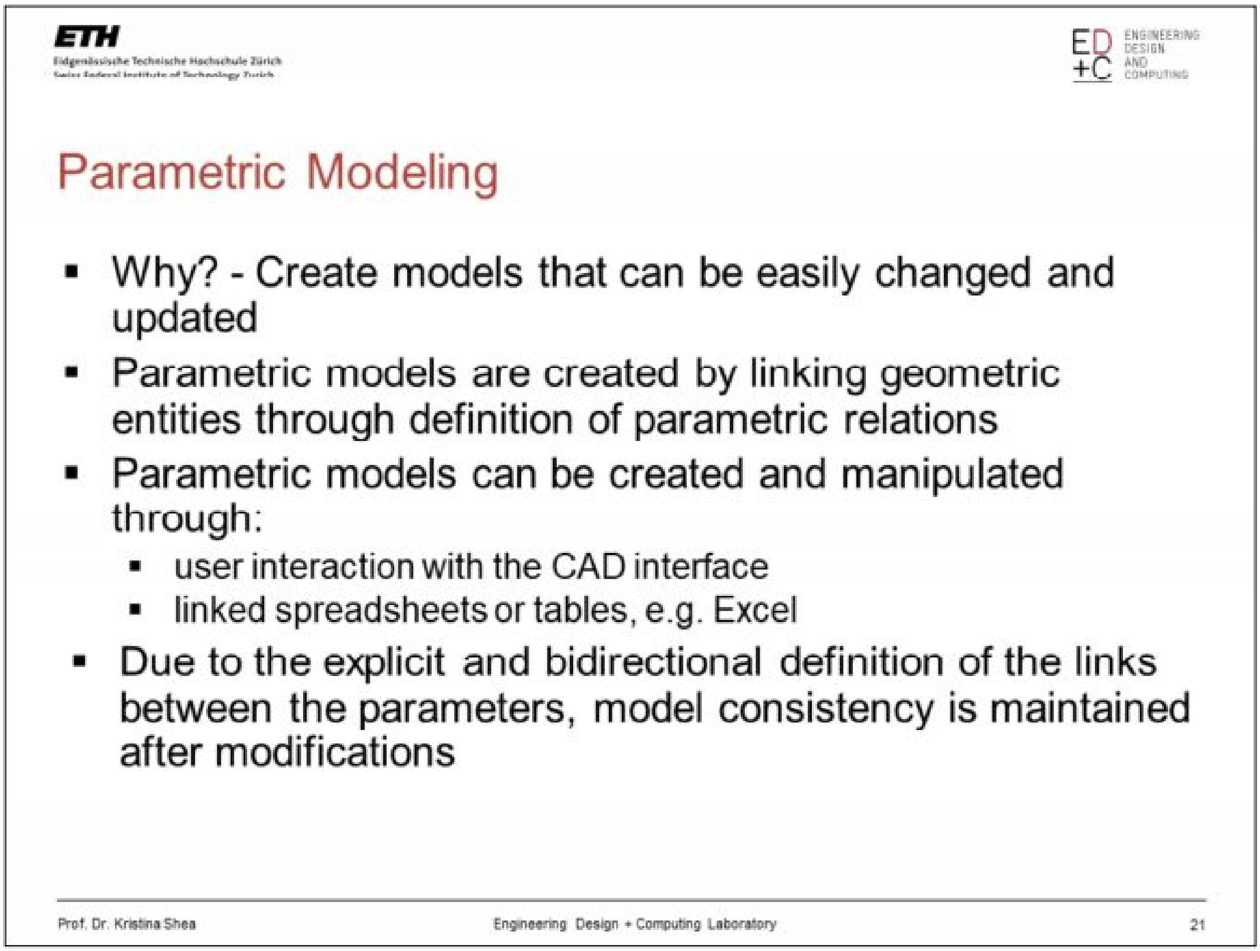

**External thread (*Aussengewinde*):**

- The thread is represented using the symbol **M** (for metric threads) preceding the dimensional value
- Major diameter (*Gewindeaussendurchmesser*): continuous **wide** line
- Minor diameter (*Gewindekerndurchmesser*): continuous **narrow** line
- End view: minor diameter shown as ¾ circle
- Hatching: extended to **major** diameter




Thread run-outs (*Gewindeausläufe*) have to be shown when necessary. The depth of the thread (*Gewindetiefe*) is represented using a wide line.

## Parametric Modeling




**Reasons for parametric modeling:**

1.Static geometry has limited possibilities for change other than deletion and creation of new geometry.

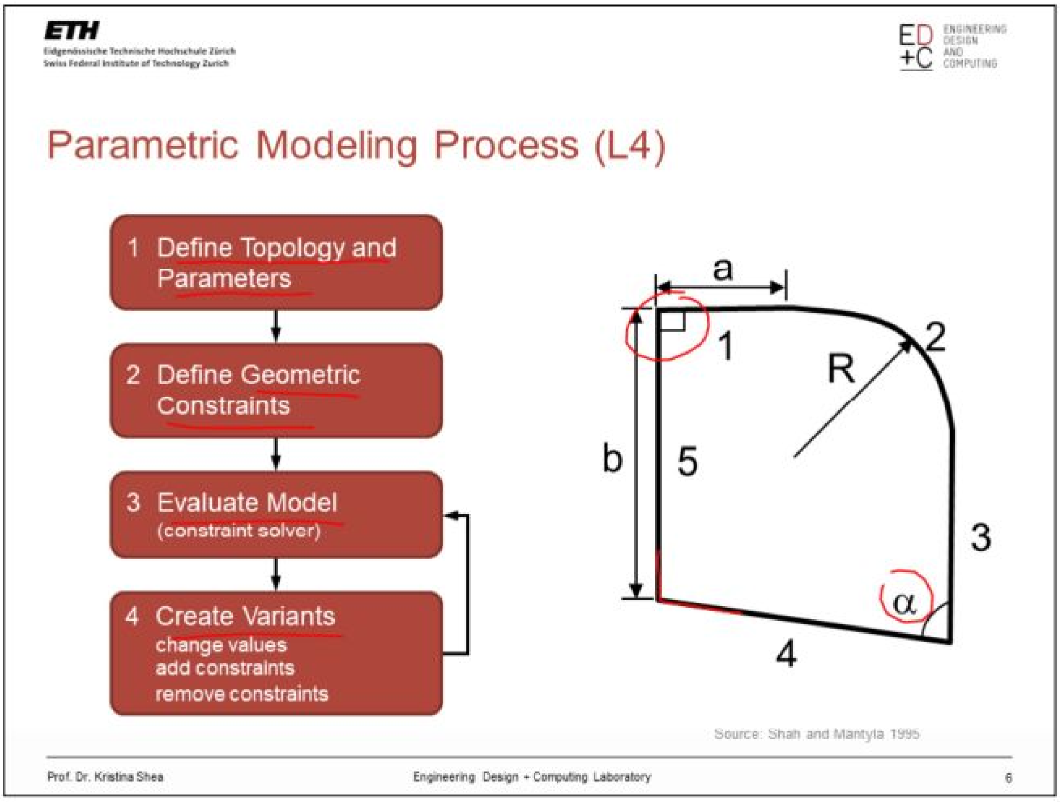
2.Design is incremental – at the beginning you don't have a solution to the design problem.

3.Most design is variational (~80%) (Shah *et al*. 1995), i.e. adapting existing designs to new requirements.

4.Potential for rapid change and updating of geometry.

5.Documentation of design intent.

6.Potential for re-use of existing CAD models (i.e. part families).



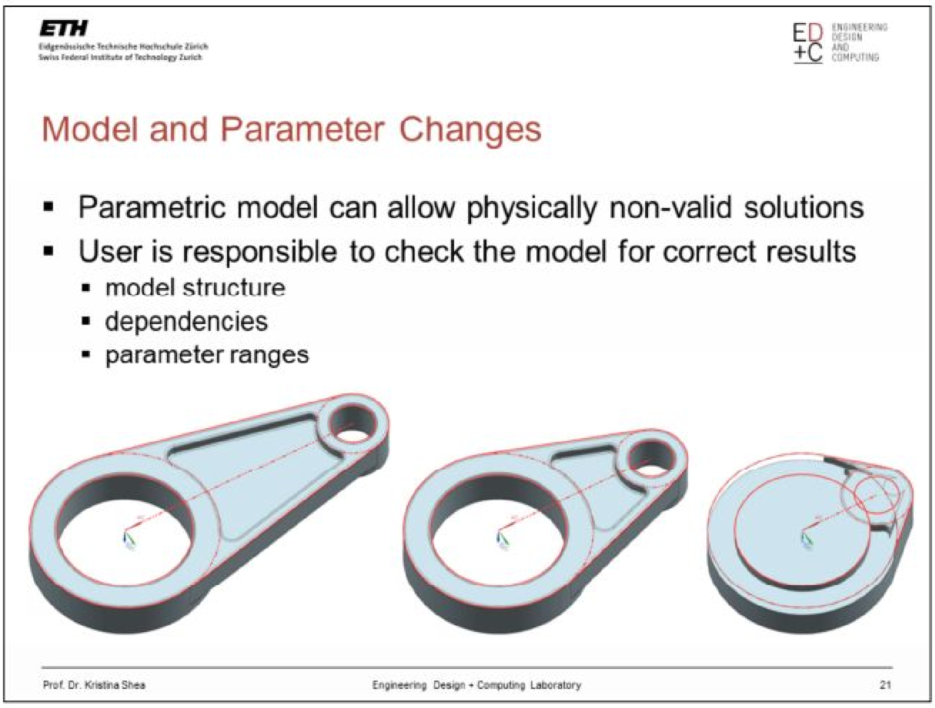

**Parametric modeling process:**

1.Using normal geometric and solid modeling – define topology and parameters

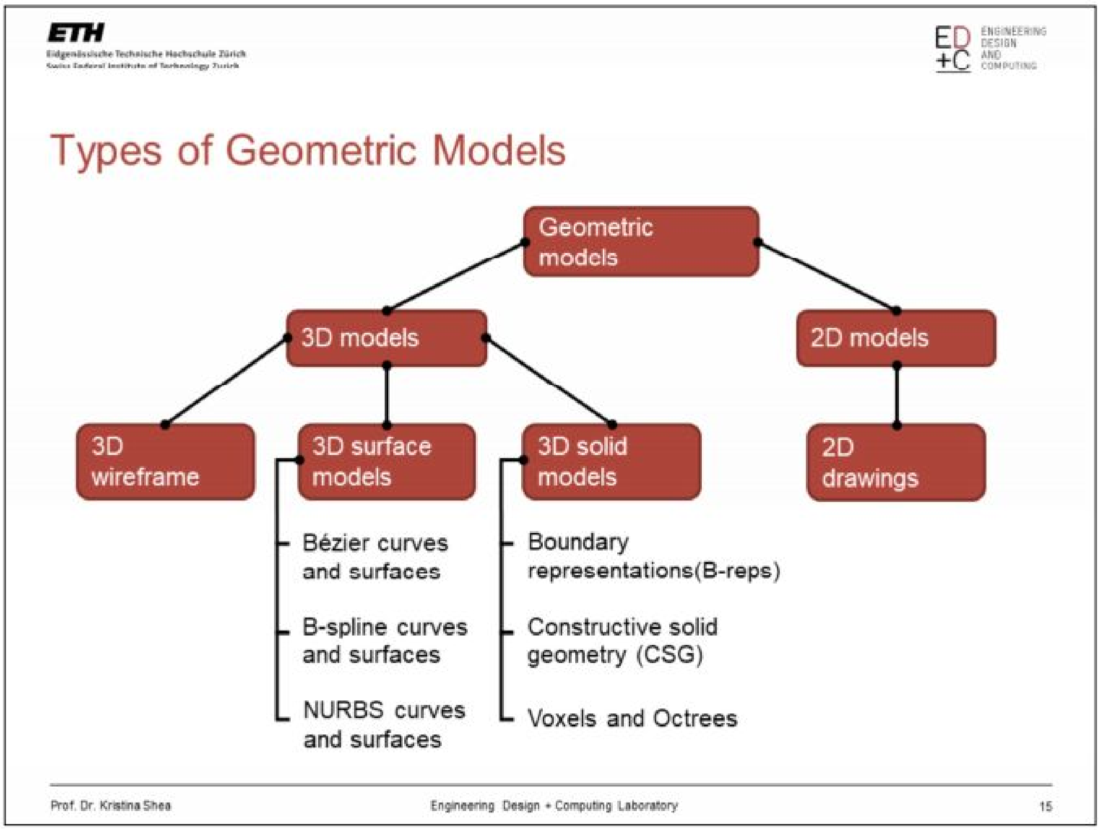
2.Define mathematical relationships, or constraints, between the parameters of the geometric model

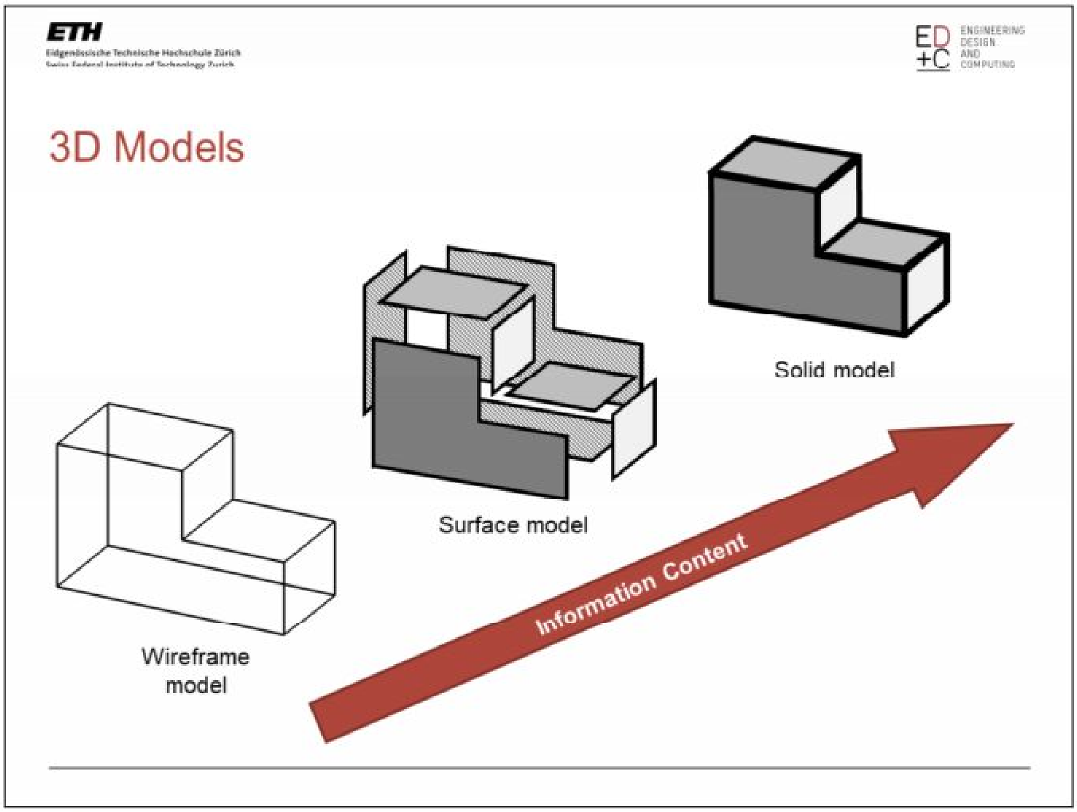
3.Solve for the dependent parameters; however sometimes it can not be solved

4.Change values of the independent parameters or the constraints themselves to create design variants.

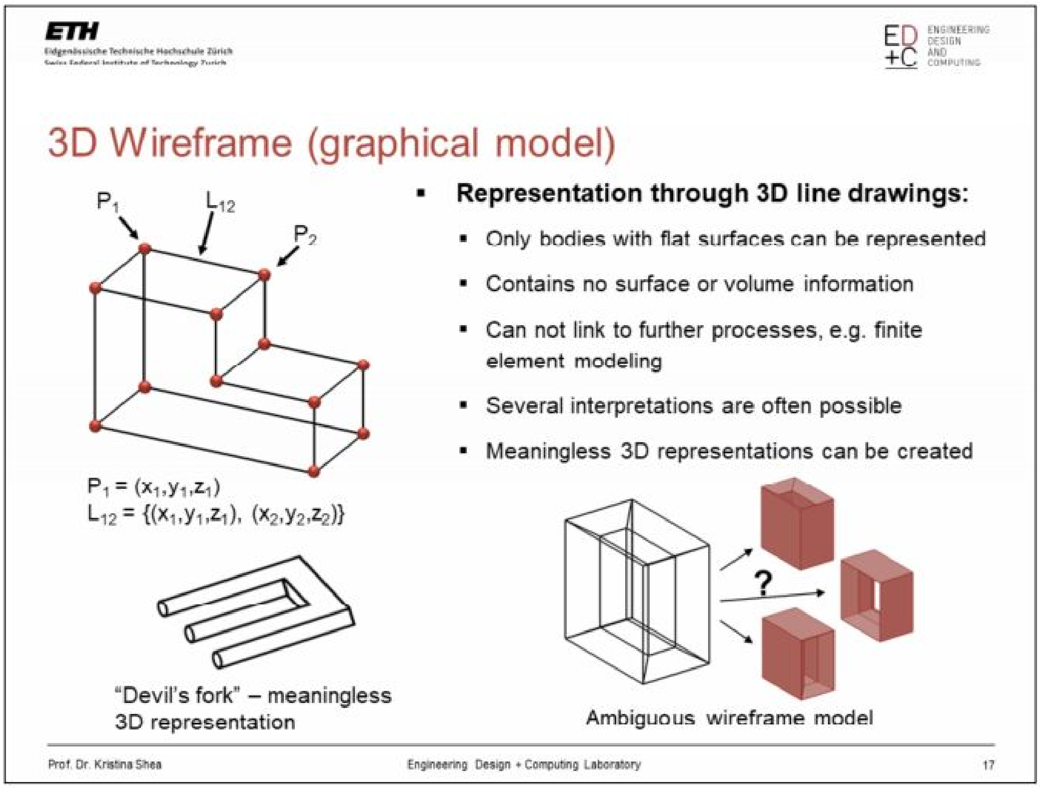


## Types of Geometric Models



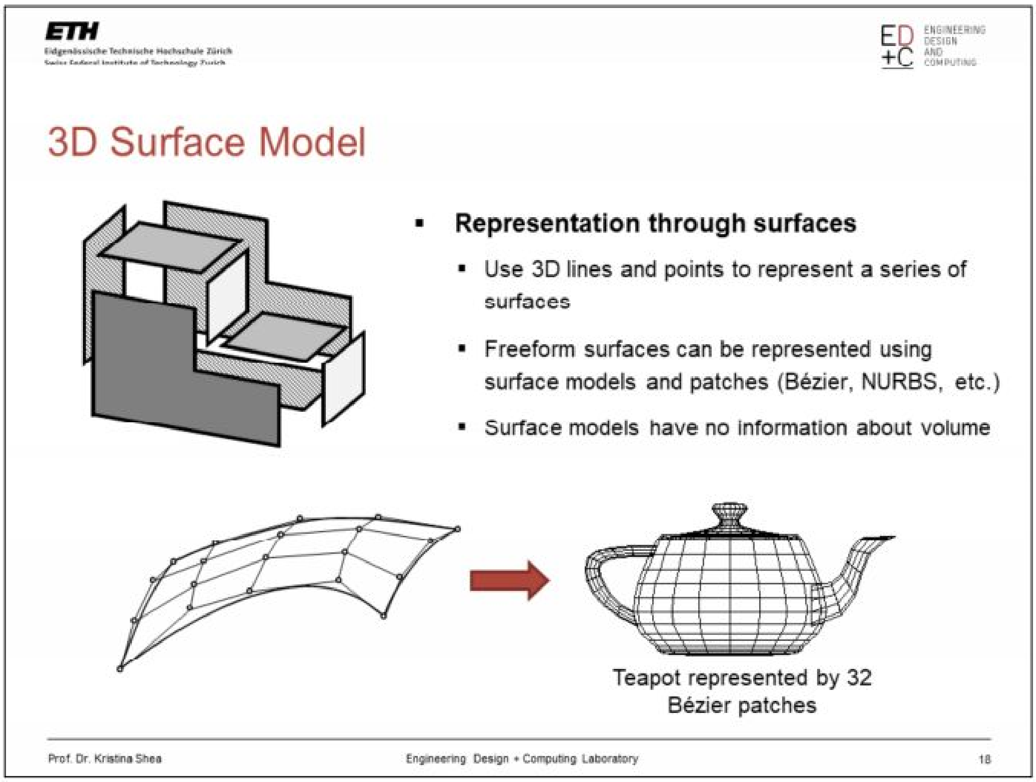




- 3D models are the most important for most mechanical engineering domains.



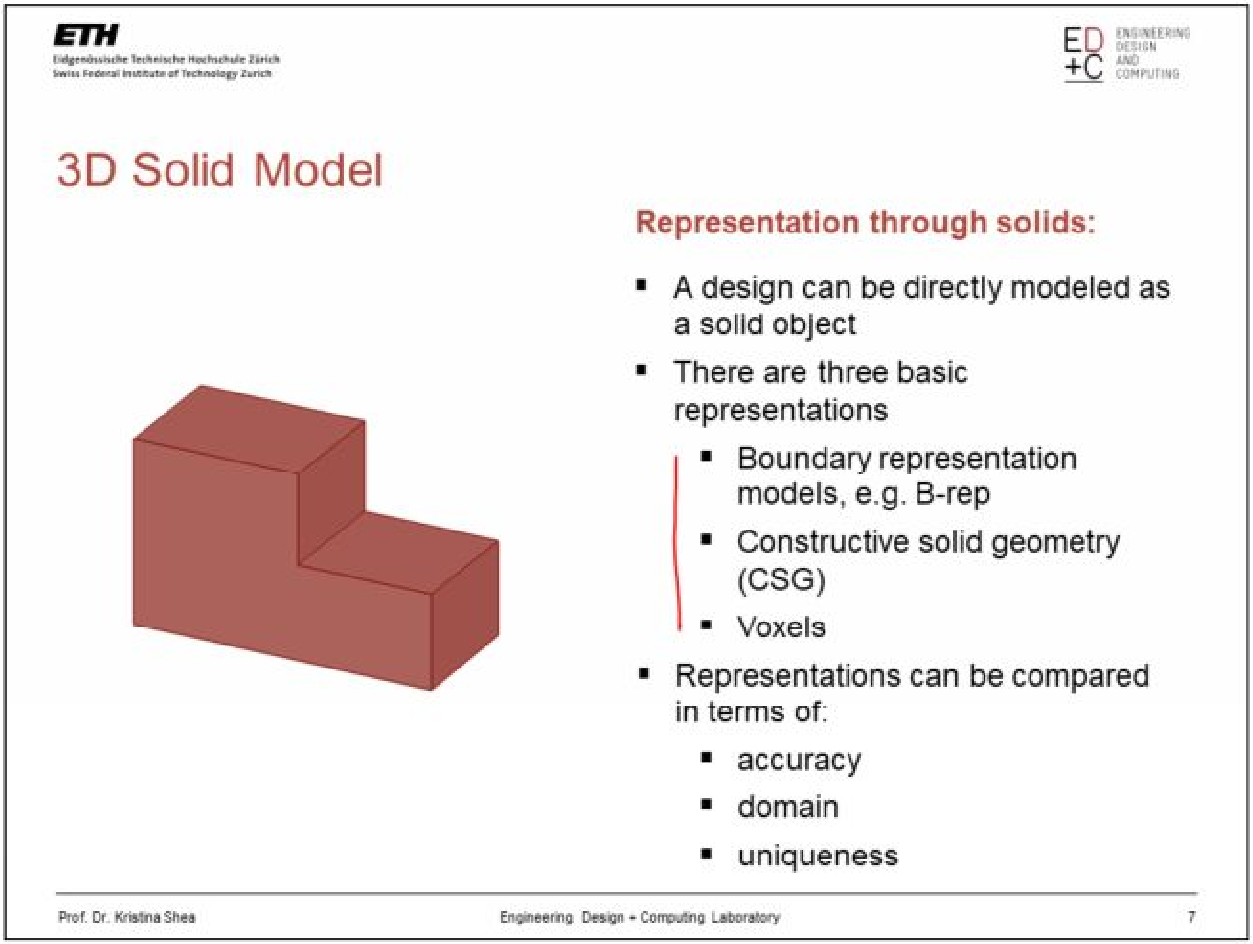

- Similar to 2D drawings
- Collection of lines and points in 3D
- Think of Escher drawings.

**Surface model**




Example: Automotive styling

**3D Solid Model**

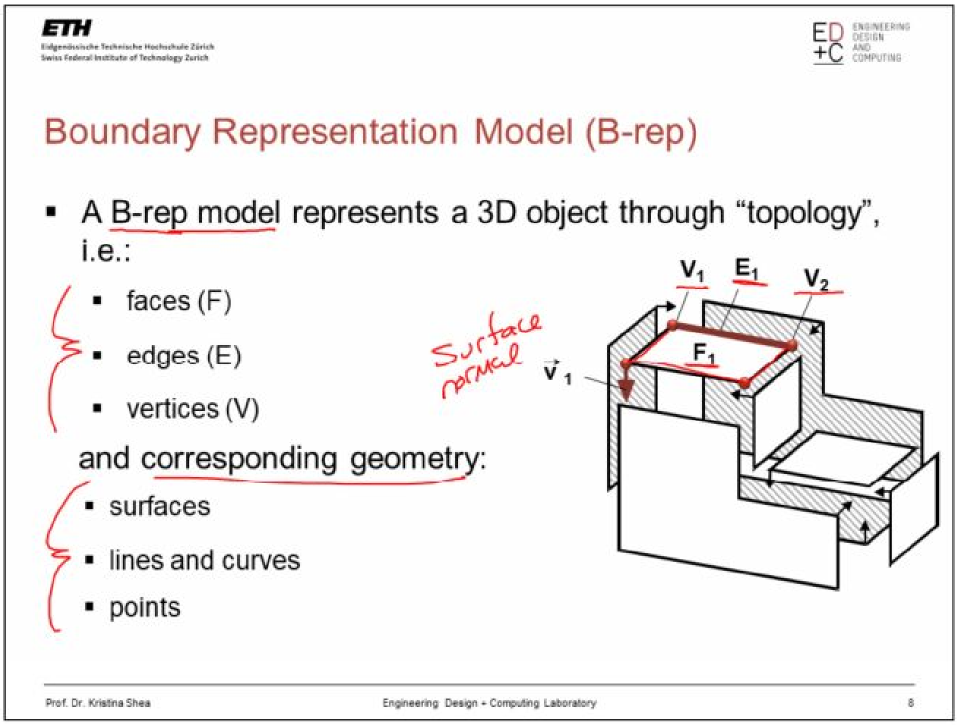



The representation of objects directly using solids was developed to address the difficulties of surface models. Solid models have the most information content of all possible model types but also the highest complexity.

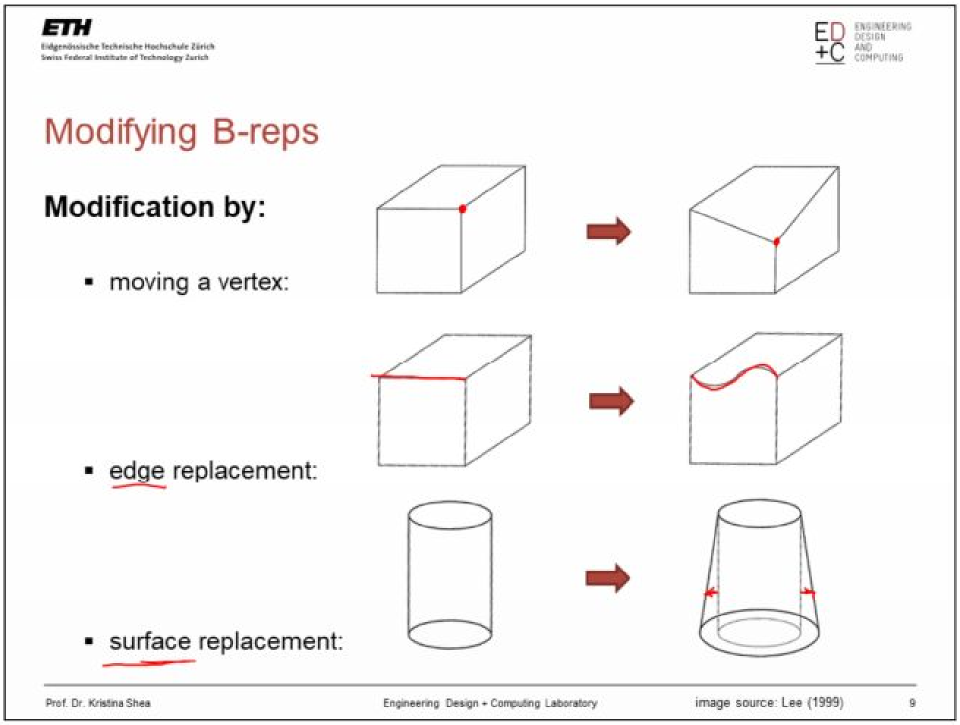
Models can be compared in terms of:

- Accuracy – how accurately does the model represent the actual 3D object?
- Domain - what range of 3D objects can be represented?
- Uniqueness - a representation is unique if it can be used to encode any given solid in only one way; unique models makes testing two objects for equality simpler

## B-rep

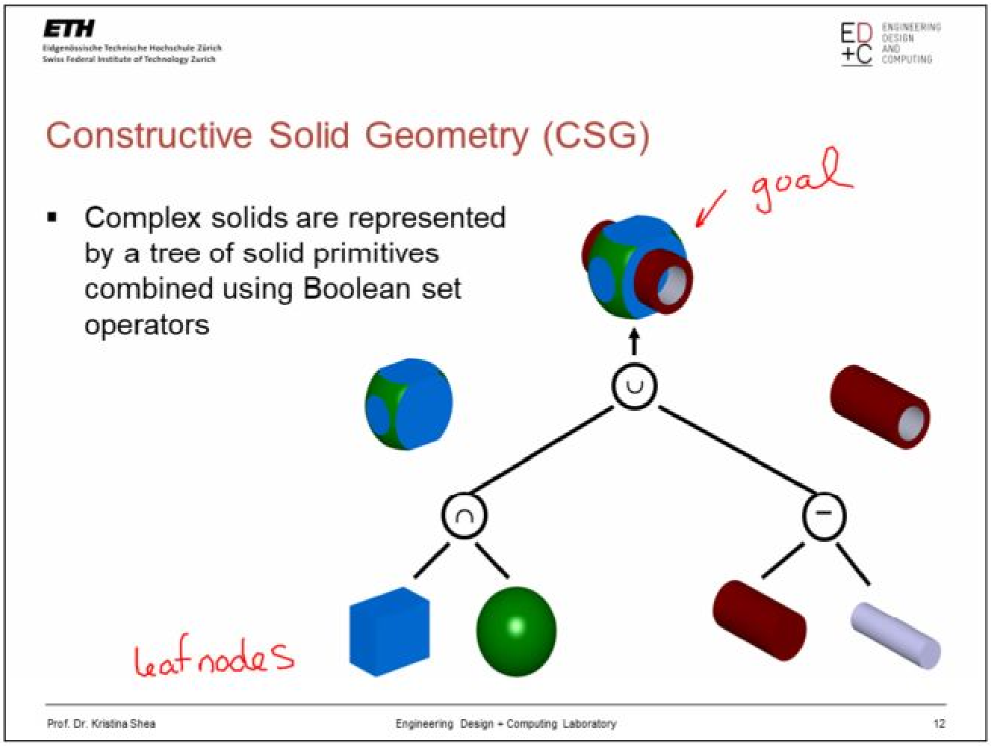



- A B-rep separates the representation of topology and geometry of a solid
- Further characteristics:
  - Accuracy – polygonal B-reps are less accurate than B-reps with curved surfaces
  - Domain – represent the widest domain of solids
  - Uniqueness – generally not unique

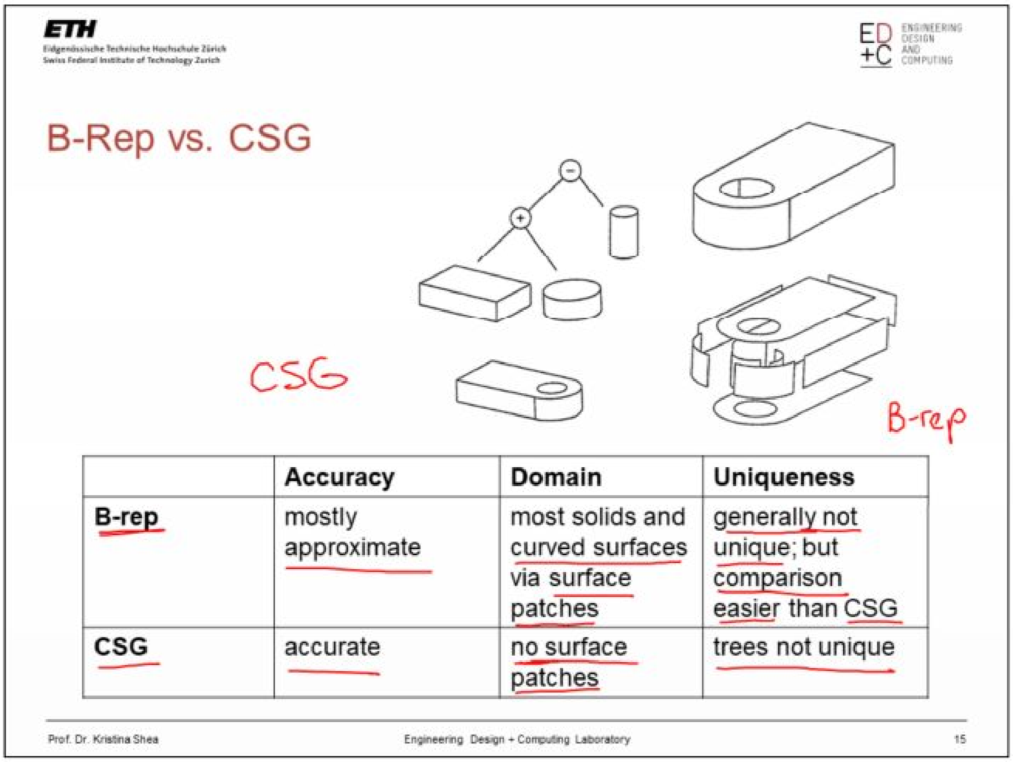



An advantage of separating topology and geometry representation is that the solid can be easily modified by replacement of the geometry alone.

**CSG (Constructive Solid Geometry)**



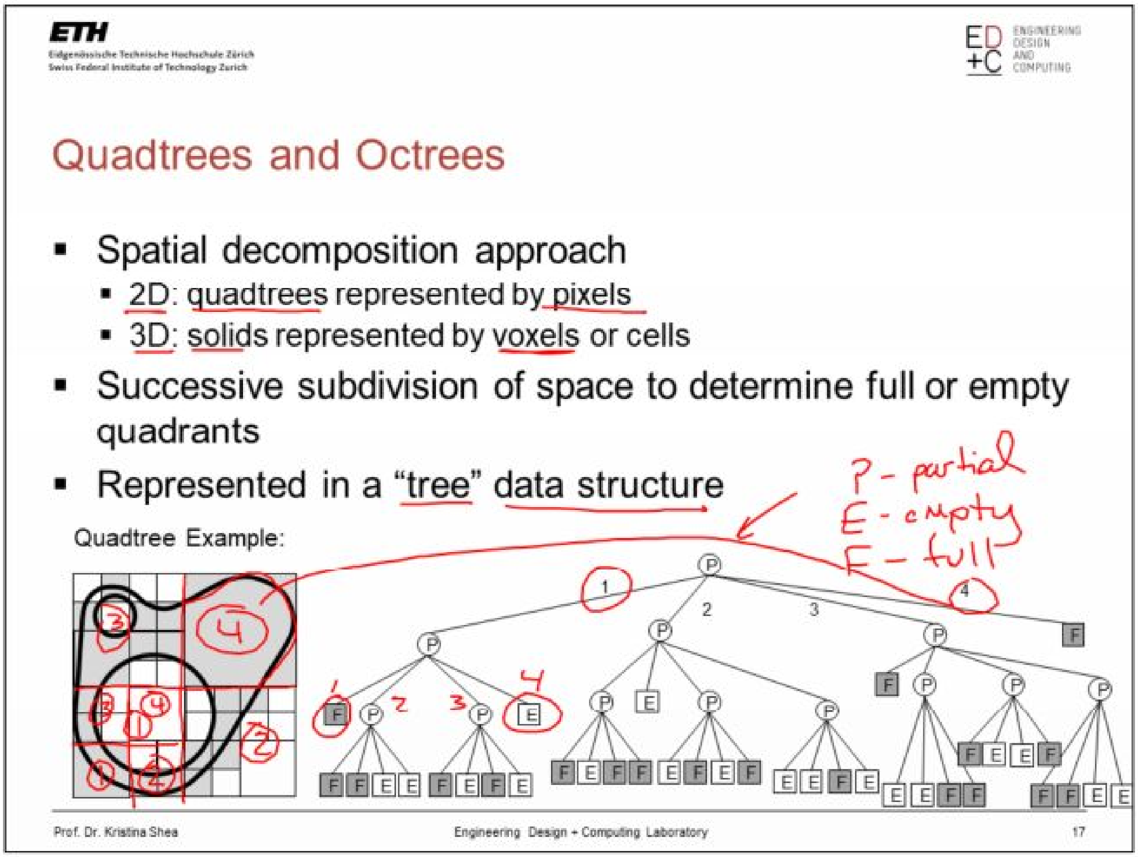

CSG - represented through a tree structure where the root nodes are the solid primitives and the interior nodes are either Boolean set operators or can also be translation or rotation operators.




| | Accuracy | Domain | Uniqueness |
|---|---|---|---|
| **B-rep** | mostly approximate | most solids and curved surfaces via surface patches | generally not unique; but comparison easier than CSG |
| **CSG** | accurate | no surface patches | trees not unique |

- Accuracy – how accurately does the model represent the actual 3D object?
- Domain - what range of 3D objects can be represented?
- Uniqueness - a representation is unique if it can be used to encode any given solid in only one way; unique models makes testing two objects for equality simpler

**Voxels**

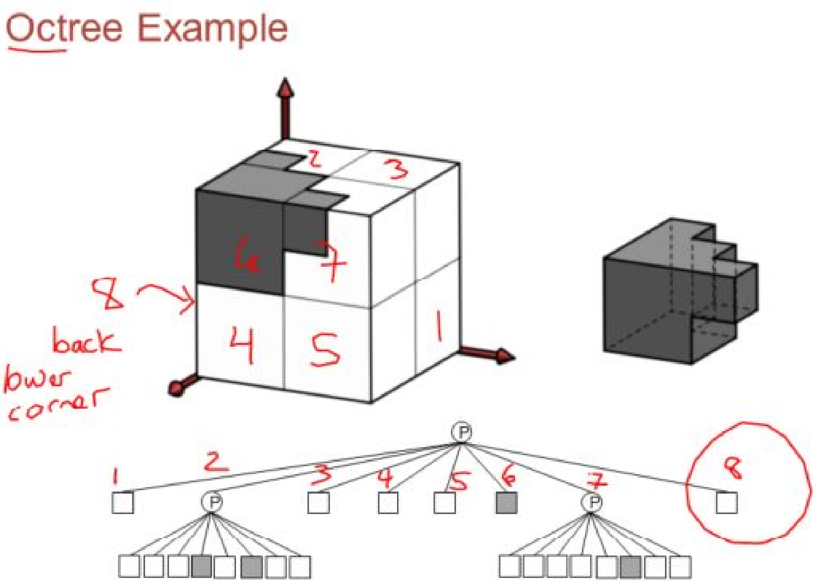


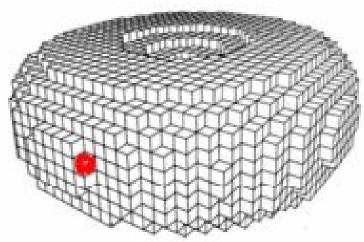

•Decomposition is another principle to represent solids.

•It is based on successive levels of subdivision

•Right is a quadtree, each time splitting each partially full space into 4 regions to determine if it is full (black), partially full (grey) or empty (white). The subdivision continues until all pixels are either full or empty.

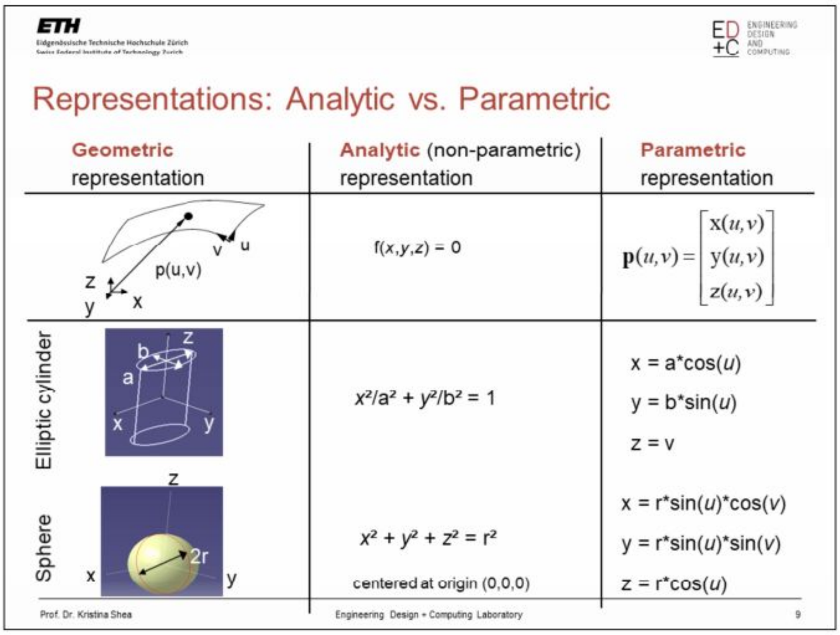
•During a boolean set operation, e.g. intersection, this representation reduces the number of comparisons compared to representing all pixels or voxels in the bounding volume (3D printing).



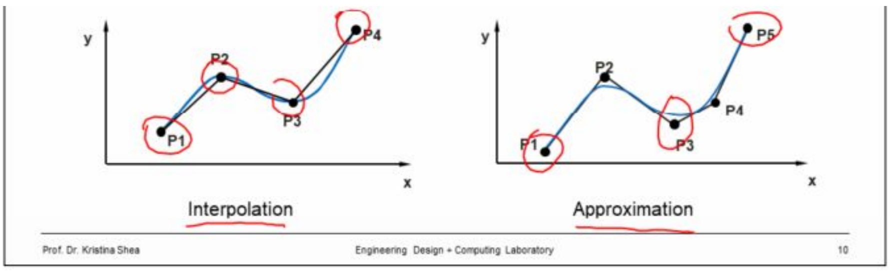
Voxel representation of torus

## Freeform Modeling

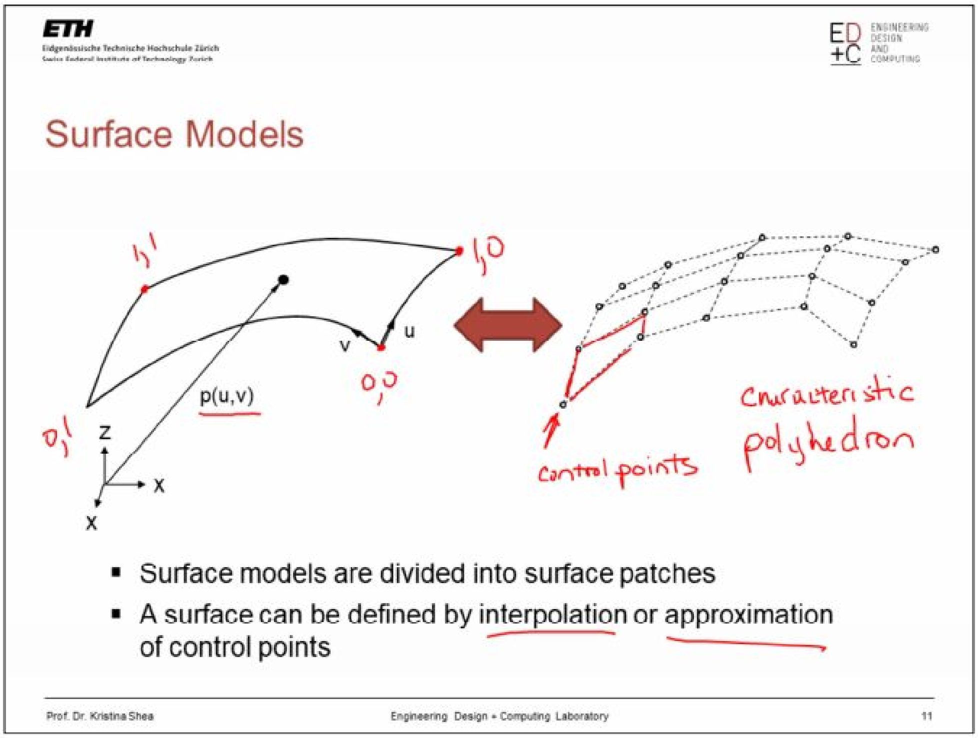



How can geometric surfaces be mathematically represented?

- Analytic not good
    - Equation is dependant on coordinate system.
    - Unless constrained, curves are unbounded.
    - Implicit form, f(x,y) =0, inconvenient to compute points on curve; find x such that y lies on curve
    - elliptic cylinder – degenerate form not dependant on z
- Parametric representation avoids these difficulties
    - u and v are generally bounded between 0 and 1.
    - Surface corners (u,v): (0,0), (0,1), (1,0), (1,1)
    - Surface edge curves (u,v): (0,v), (u,0), (1,v), (u,1)

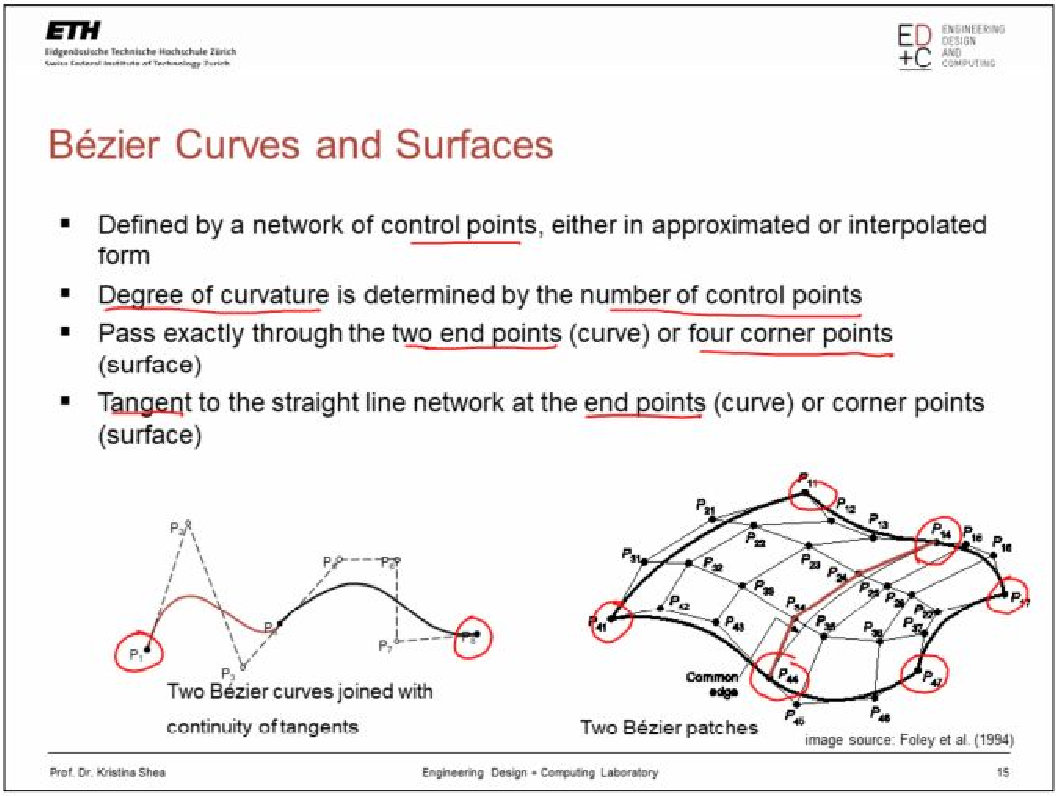



- A numerical curve (or surface) is defined by a set of control points.
- These control points can be either interpolated or approximated.
- The linear interpolation between control points is called the characteristic polygon.
- The figures on the bottom give a simple example of the difference between interpolation and approximation for a curve. As shown, an interpolated curve is coincident to all control points whereas an approximated curve is coincident with only a few control points (on the right: P1 and P5) and approximates the others.



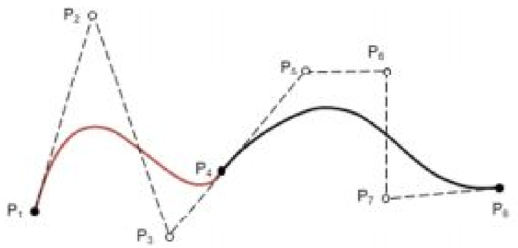

- To model a more complex, or freeform surface, requires dividing the intended geometry into a set of surface patches. Each surface patch consists of a set of control points. The polyhedron connecting all control points with straight lines is called the characteristic, or control, polyhedron. For curves, this is called the characteristic polygon. A surface is then defined either by interpolation or approximation of the control points.

### Bézier Curves



## Disadvantages of Bézier Curves and Surfaces

- Degree of curvature is determined by the number of control points, e.g.
  - two control points create a linear curve (1st order polynominal)
  - three control points create a quadratic curve (2nd order polynomial), etc.
- Calculation effort increases with the number of control points
- Changing the location of any control point affects the shape of the whole curve.

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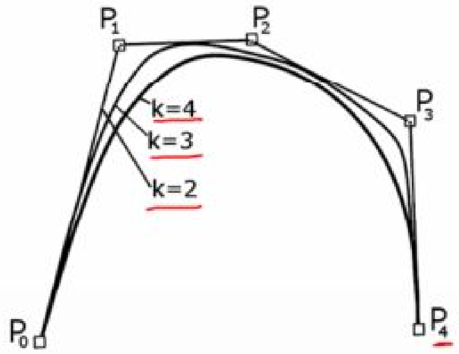
### B-spline Curves and Surfaces

ETH
Eidgenössische Technische Hochschule Zürich
Swiss Federal Institute of Technology Zurich

ED+C Engineering Design and Computing

## B-spline Curves and Surfaces

- Generalized form of Bézier curves and surfaces
- Defined by a network of control points and degree of curvature, $k$-1, for better local control




Influence of degree, ($k$-1), on B-spline

image source: Shah and Mäntylä (1995)

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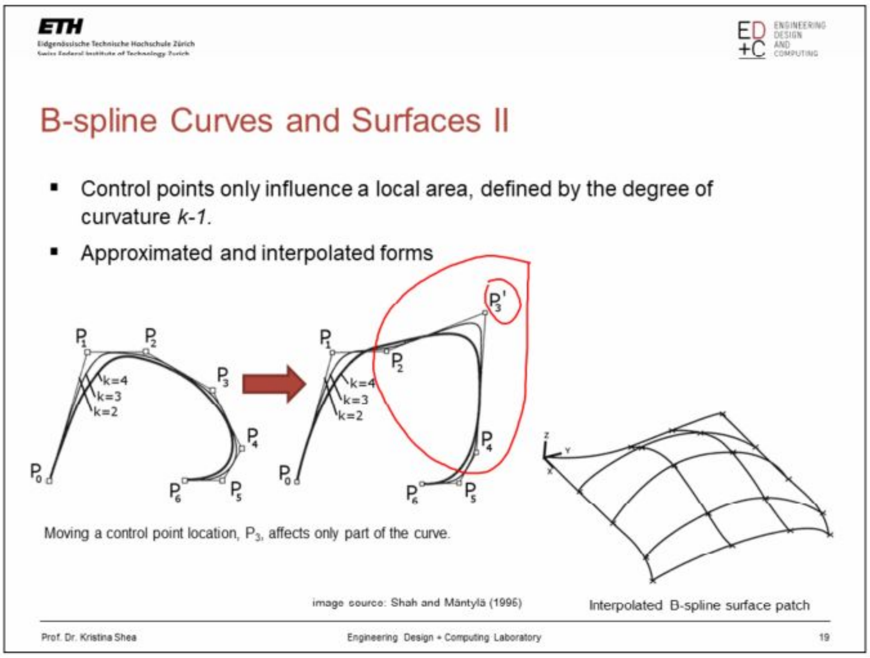
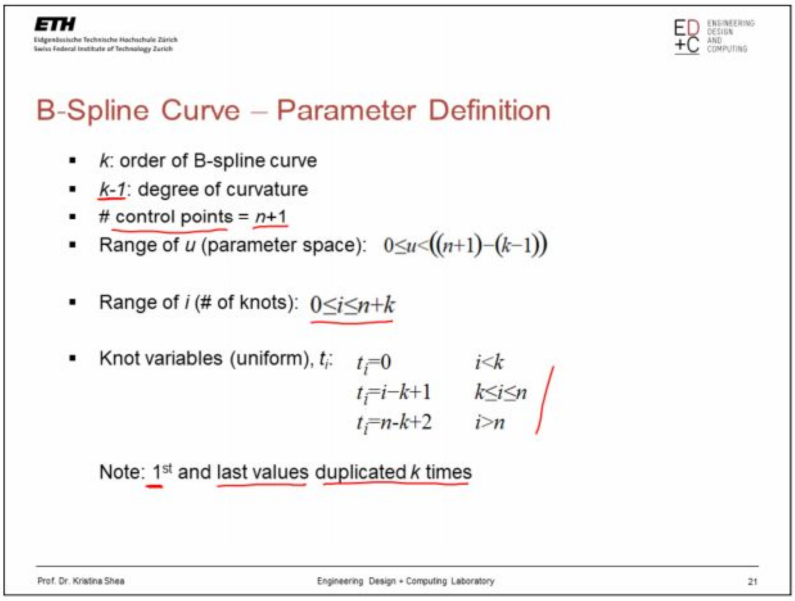


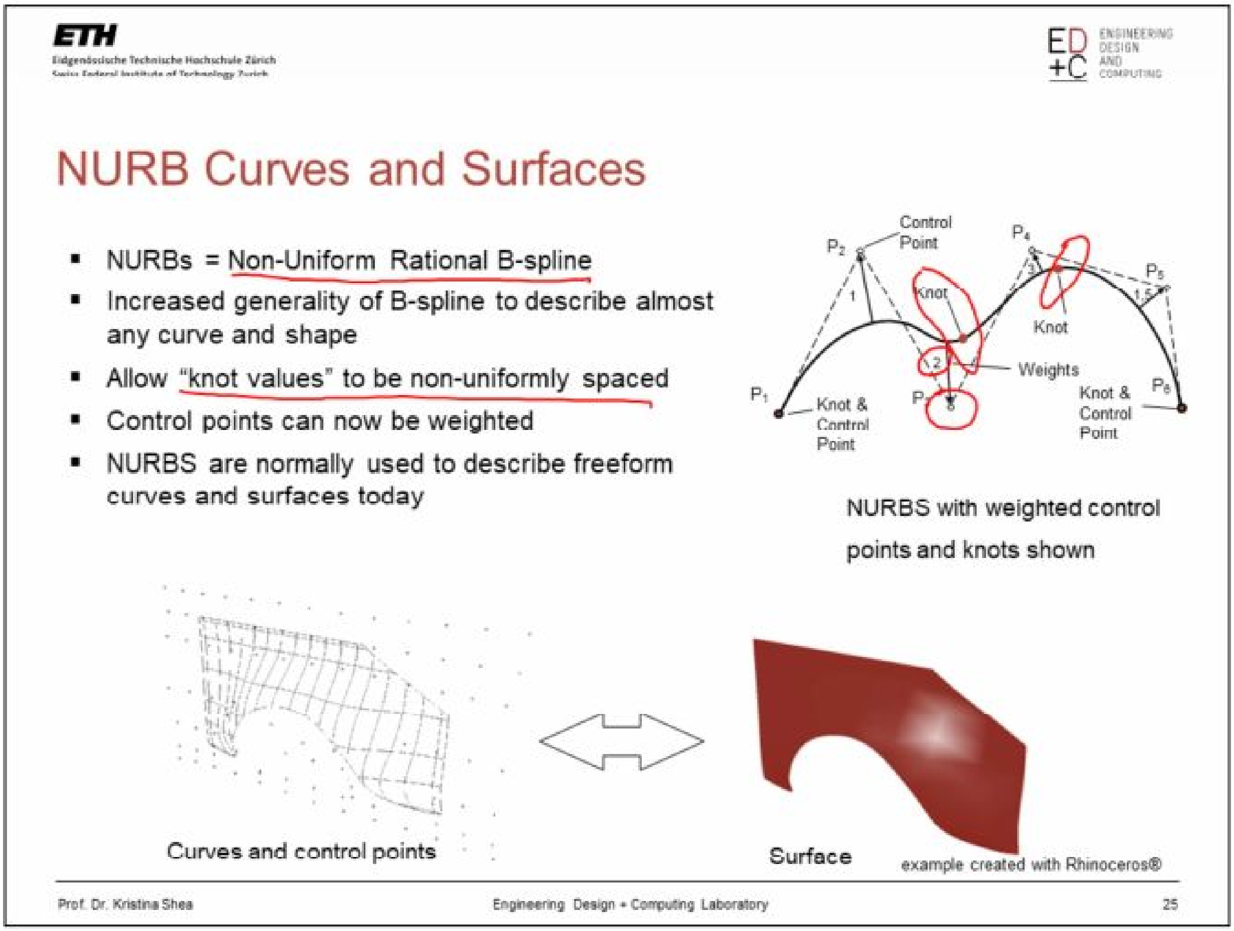

Through the addition of *k*, contrary to Bézier curves, the control points influence only a local area. The size of the influence area is defined by the degree of curvature, as can be seen on the left. Consider *k* = 2. One can see that through the movement of $P_3$ to $P_3'$, only two curve segments are affected, the curve between $P_2$ and $P_3'$ and the one between $P_3'$ and $P_4$. Further, the number of control points influences but does not fully control the degree of the curve or surface.

Just like with Bézier curves and surfaces, approximated and interpolated forms of B-Splines are possible.



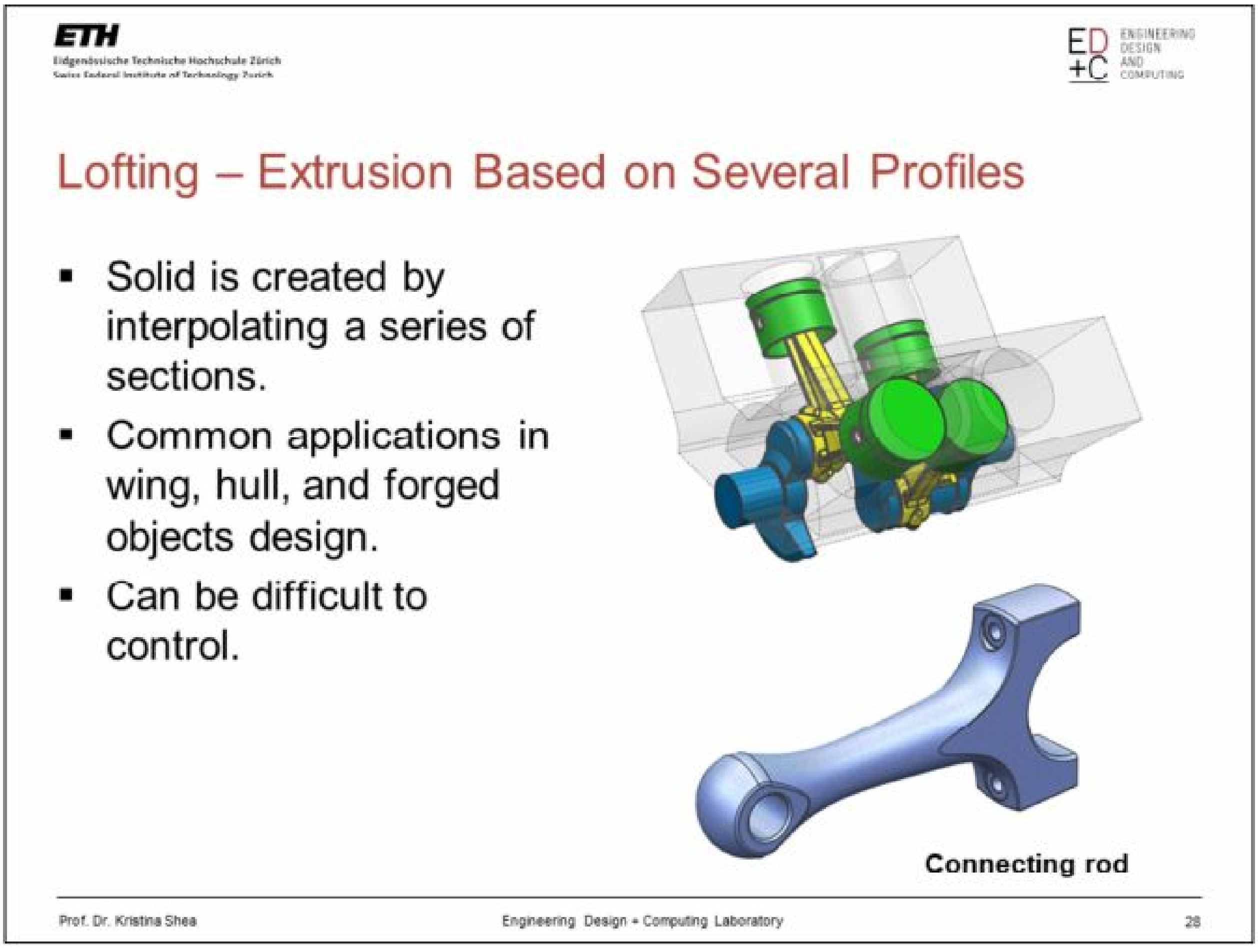

- The order *k* of the curve must be equal to or greater than the number of control points.
- Formula for u results in a parameter range of 0<=*u*<=1 for case when *n*+1 = *k*, i.e. Bézier
- Knot variables
  - divide a curve into segments
  - relate the parametric variable u to the control points
- Gap between uniformly spaced knots is always 1.
- Uniform spaced knots result in duplicated 1st and last knots *k* times that results in the curves passing through the first and last points.

**NURB Curves and Surfaces**




- The abbreviation NURBS stands for Non-Uniform Rational B-Spline. NURBSs have an increased generality compared to B-Splines and thus provide a general representation for both B-Splines and Bézier. The main advantage of NURBSs is their ability to accurately describe almost any curve and shape. These properties lead to flexible and at the same time accurate models. This is why NURBSs are the standard in many CAD systems to describe freeform curves and surfaces.
- As with B-Splines, a NURBS curve, as shown in the upper right, is defined by a set of control points and a degree of curvature parameter. Typically, as with B-Splines, the curve is an approximated form with point continuity ($0^{th}$ order) at the endpoints.
- Different from B-Splines, NURBS allow for non-uniformly spaced knots. Further, a weight is used for each control point that provides an indication of the attraction of the control point to the curve. The larger the weight, the more the curve is pulled toward the associated control point. (See movie.)

## Lofting

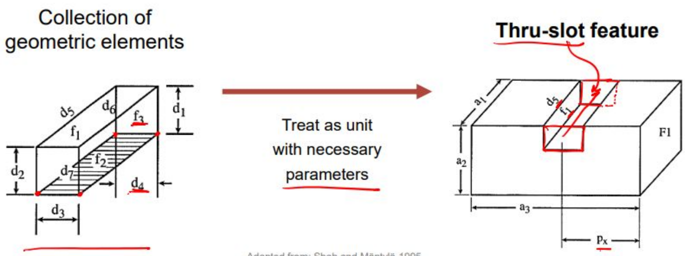



- **In NX:** Lofting = Through Curves

# Feature-Based Design

## Motivation for Feature-Based Design (FBD)

- Easy geometry creation
- Fast propagation of design changes
- Building-in the feature information in the model



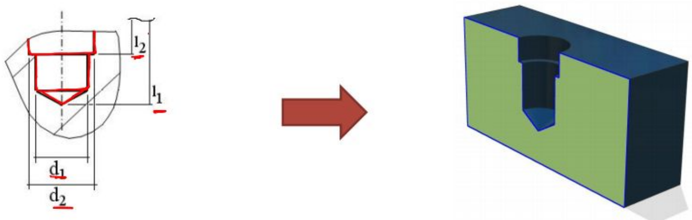

Adapted from: Shah and Mäntylä 1995

## Introduction - Limitations of Geometric Modeling

- Geometric construction methods are often too low level
- Geometric models only have one level of abstraction (dimensions, parameters and constraints)
- Design changes can be time consuming
- Little capture of design intent, apart from parametric relations between low-level geometry
- Not enough information for many downstream applications, e.g. manufacturing

## What is a feature?

- **Definition**: A feature represents the engineering meaning, or semantics, of the geometry of a part or assembly.

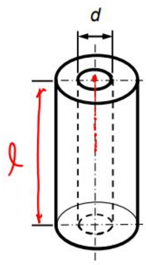


Stepped blind hole

Cross-section cut of a block showing a stepped blind hole feature

## Dependent (inter-feature) properties

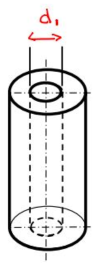
- Derived parameters and dimensions , $l$
- Feature location , co-axial cylinder
- Feature orientation
- Constraints on feature size, location, or orientation parameters
- Inter-feature tolerances




Thru-hole in a cylinder

## Independent (intra-feature) properties

- User-defined parameters and dimensions , $d$
- Geometric shape , cylinder
- Parameter labels or names , "thru-hole"
- Derived parameters and dimensions (user-defined)
  e.g. $d_2 = .6d$, (stepped hole)
- Form and size tolerances
- Orientation tolerances

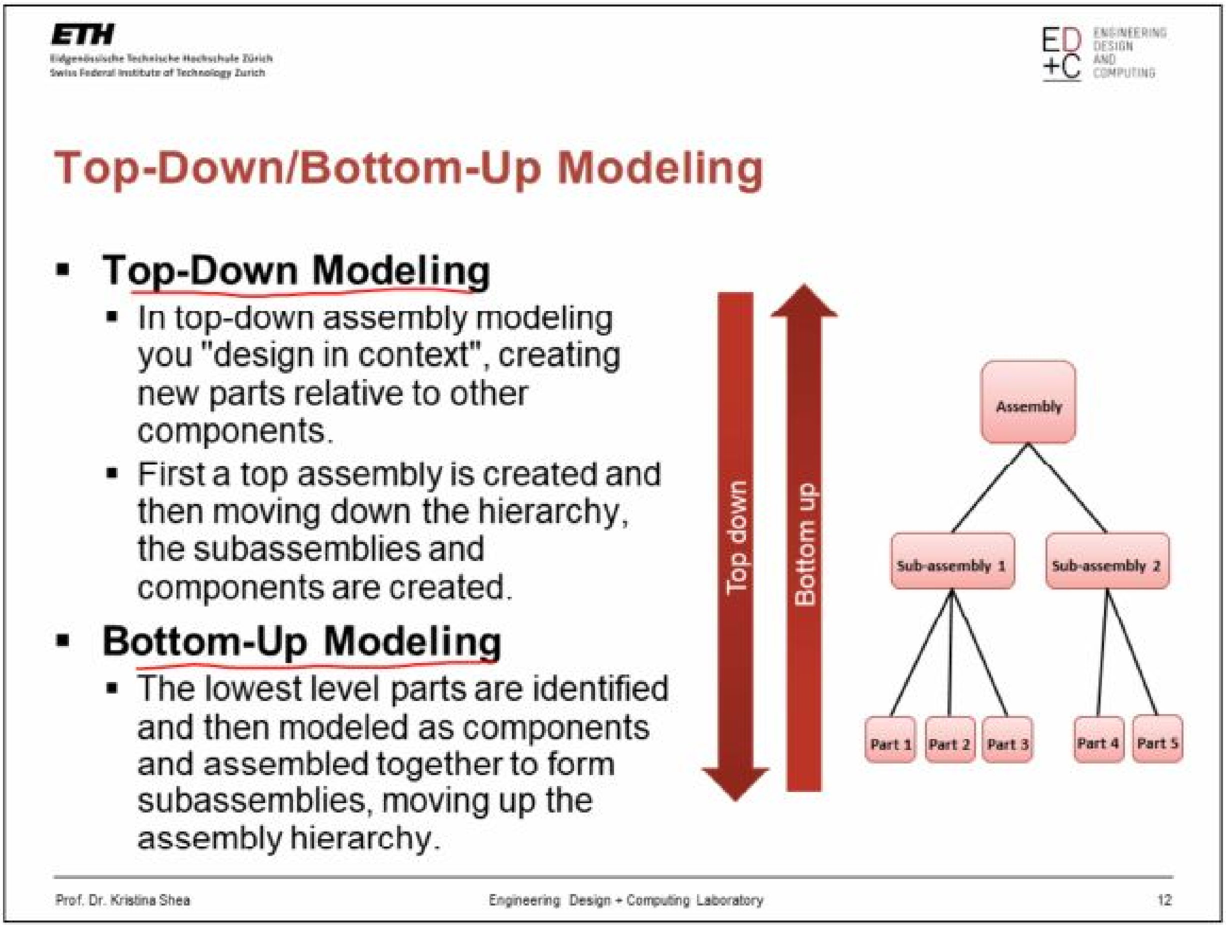



Thru-hole in a cylinder

**Fused-Deposition Modeling (3D-Druck)**

- Fused-Deposition Modeling (FDM) is an additive manufacturing process that builds parts and assemblies through extrusion of plastic materials, including support structures.
- The digital design-to-fabrication process starts from a CAD model, which is converted to a .STL file, a build file and finally 3D printed parts.
- Design for Additive Manufacture (DfAM) provides guidelines and constraints on how to design and model parts such that they can be printed robustly on a 3D printer.

## Assembly


ETH
Eidgenössische Technische Hochschule Zürich
Swiss Federal Institute of Technology Zurich
ED +C ENGINEERING DESIGN AND COMPUTING
Top-Down/Bottom-Up Modeling
Top-Down Modeling
In top-down assembly modeling you "design in context", creating new parts relative to other components.
First a top assembly is created and then moving down the hierarchy, the subassemblies and components are created.
Bottom-Up Modeling
The lowest level parts are identified and then modeled as components and assembled together to form subassemblies, moving up the assembly hierarchy.
Top down
Bottom up
Assembly
Sub-assembly 1
Sub-assembly 2
Part 1
Part 2
Part 3
Part 4
Part 5
Prof. Dr. Kristina Shea
Engineering Design + Computing Laboratory
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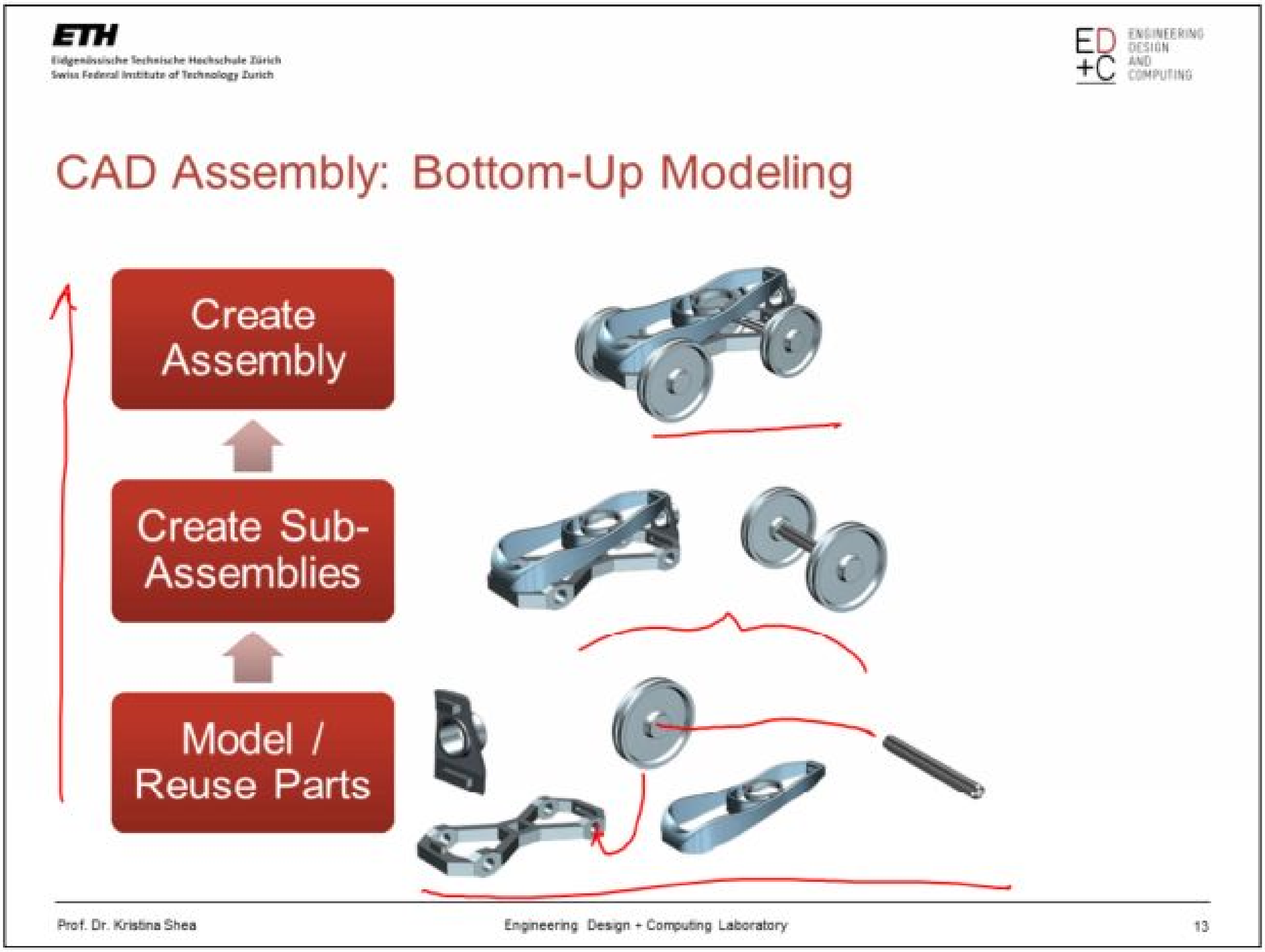



**Bottom-Up** Modeling means that single parts or sub-assemblies already exist or are created and can be used to build the main assembly.

This is usually used for adaptive design (*Anpassungs- und Änderungskonstruktionen*), where the product structure is already known.

+ CAD parts are independent from each other
    - Can be created in parallel (multiple developers)
    - Clear total complexity of the CAD model

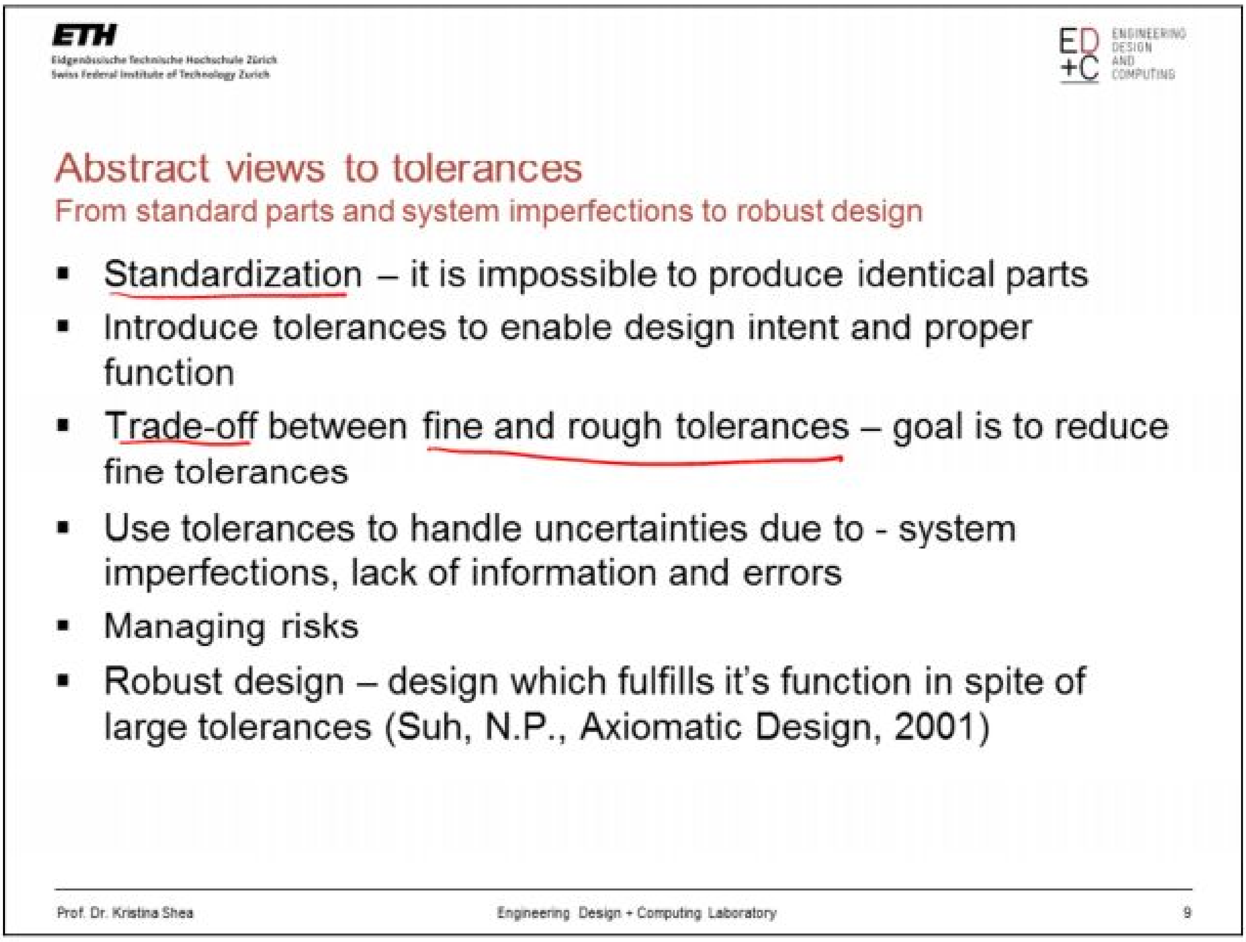
- No representation of real dependencies between the parts in the CAD system
    - Despite the real components being geometrically dependent on each other, e.g. a shaft diameter and the corresponding hole diameter

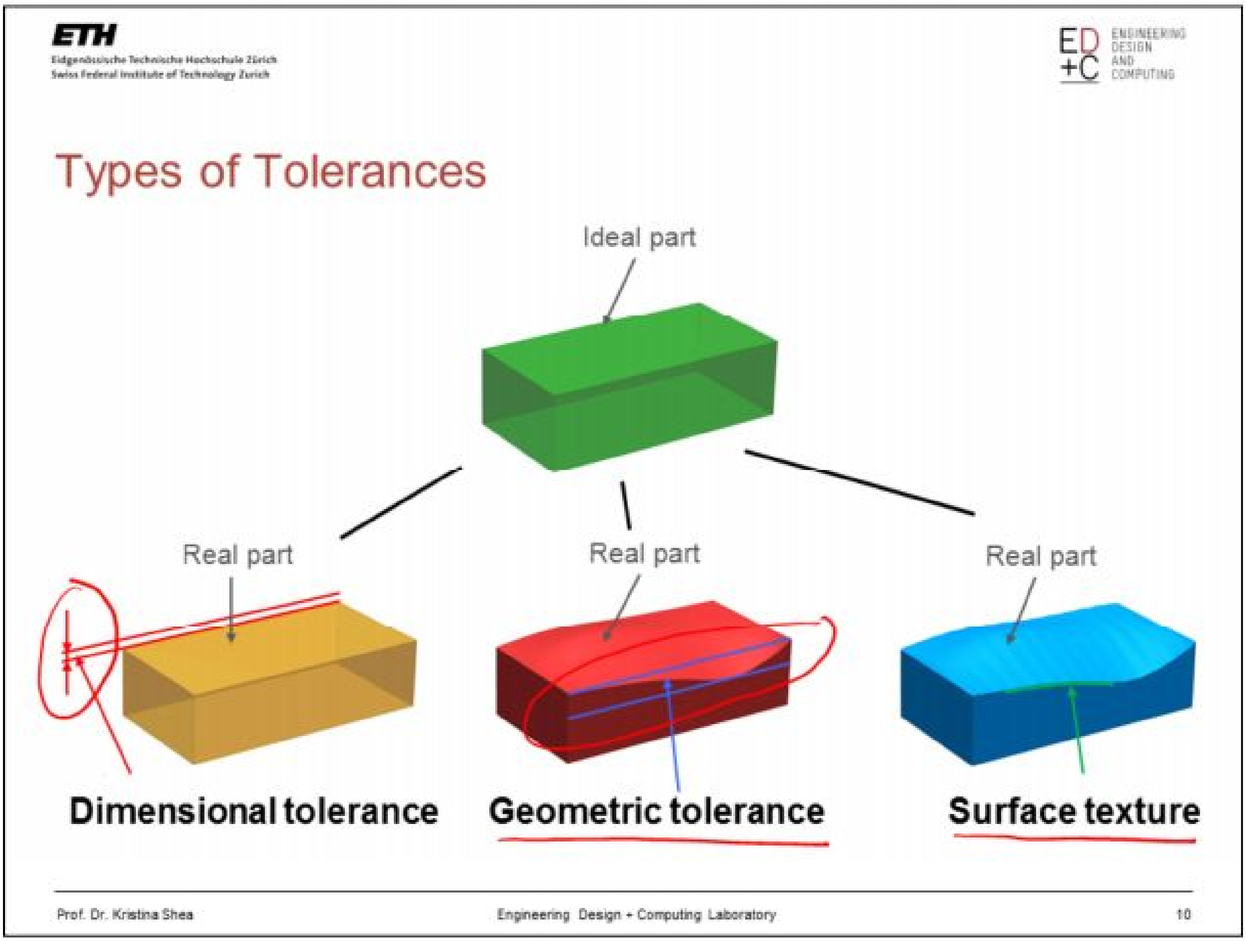
- High effort for implementing changes
    - All parts have to be adapted individually
    - Afterwards it is necessary to re-check the assembly for functionality, collisions and interaction of the parts

Bei Assembly-Cuts werden unterschiedliche Teile mit unterschiedlichen Abständen oder Richtung der Schraffur kenntlich gemacht.

## Toleranzen

Alle Vorlesungen zu Toleranzen werden ausgezeichnet im Technischen Taschenbuch, Schaeffler, Aufl. 2017, Seiten 371 - 431






Real parts are never exact and never identical. There is always a **deviation from the intended dimension**.
Tolerances define the allowed deviation from the nominal value.

Tolerances are needed to:

- Allow **interchangeability** of parts e.g. to repair a machine or for maintenance
- Guarantee **function** of the whole system

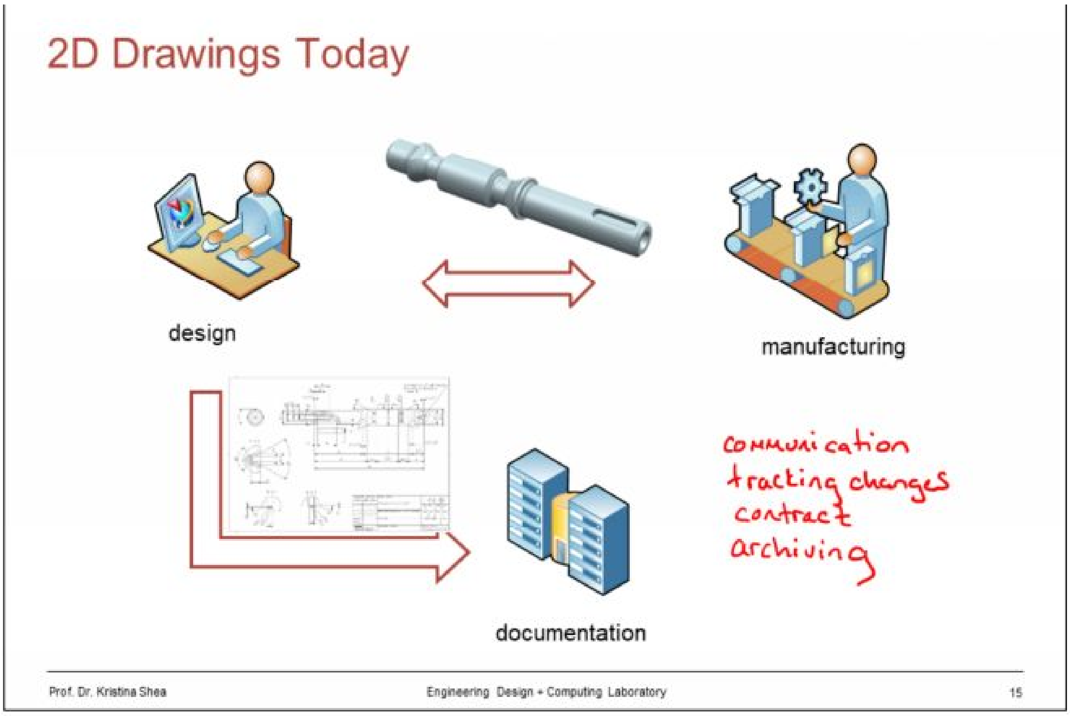
The deviations can be described by three categories:

- Dimensional tolerance (*Masstoleranz*)
- Geometric tolerance (*Form- und Lagetoleranz*)
- Surface texture (*Oberflächenangaben*)

## Oberflächenbeschaffenheit

Technisches Taschenbuch, Schaffler, Auflage 2017, Seiten 357 - 370

# 2D-Drafting




Today, design and manufacturing are mostly based on digital models. The communication between design and manufacturing departments is often through exchange of digital information. Technical drawings (2D) however are still currently inevitable.

Why are 2D drawings still used?

- 2D drawings are used as a means of communication, e.g. with clients
- Changes in 3D models are hard to track
- States in the development process are documented using drawings (released drawings get a unique drawing number and every revision gets a unique revision number)
- A physical copy of a technical drawing can be signed by several persons, i.e. it can be used like a contract people sign if they agree with it
- Archiving of designs is still sometimes done using 2D drawings to avoid issues with CAD version changes.

**Advantages and Disadvantages of 3D Models:**

Advantages:

- Exact model
- Derivation of CNC (Computer Numerical Control) programs for machining parts
- Possibility to derive tools (e.g. forms for cast parts)
- Changes in the model are associated with the drawing, i.e. a change in the model will change all views in the drawing

Disadvantages:

- Commonly no tolerances or properties, e.g. material, of the part are defined in the model

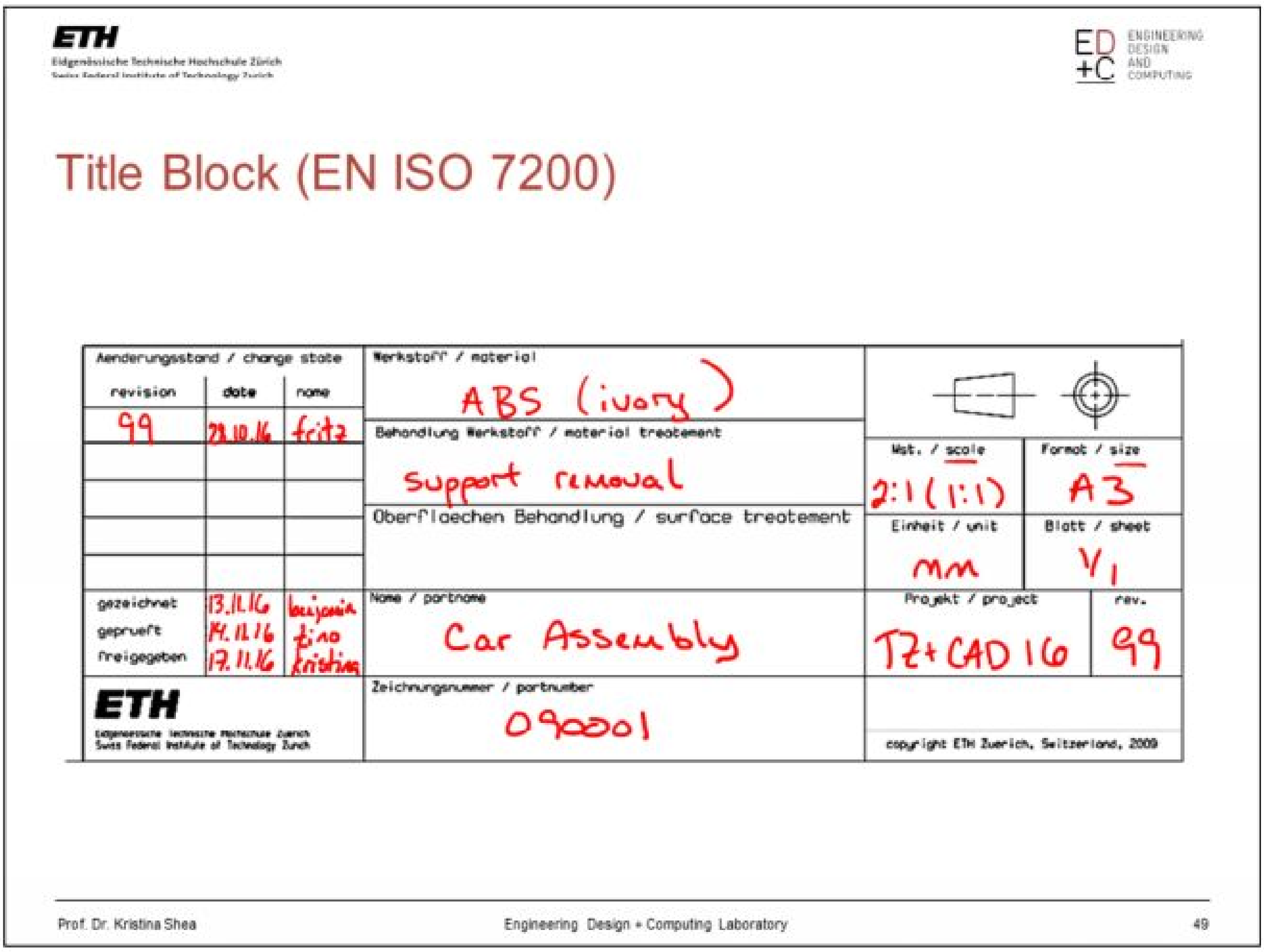
**Advantages and Disadvantages of 2D Drawings:**

Advantages:

- Means of communication between design and manufacturing (ONE language worldwide)
- Contains ALL relevant information for manufacturing a part
- Is a binding document
- Information is accessible without CAD system

Disadvantages:

- Derivation of 2D drawings is time consuming




**Material and surface treatments** (mostly used with only one object or all parts are the same):

- Project number or name
- Requirements to consider: e.g. strength, stiffness, weight, usability of the material for certain manufacturing processess, costs
- Material treatments: e.g. annealing, artificially aging, quenching, tempering
- Surface treatments: improvement of corrosion properties (chrome-plating, galvanizing), wear properties (surface hardening, hard anodizing)
- Optical properties: anodize or eloxadize with color, polishing

**Designation of the object:**

- Title / part name (serves as the first recognition feature of the drawing.)
- Identification number / part number (every part needs its own, unique and unambiguous number for its identification, since usually its name cannot be unambiguous.)
- Project number or name

**Drawing status:** (“Designed by”, “Checked by”, “Approved by”)

- A drawing is a binding document, that has to be signed and dated.
- Revision number (an already approved drawing gets a new revision number (e.g. 01, B) each time it gets changed.)

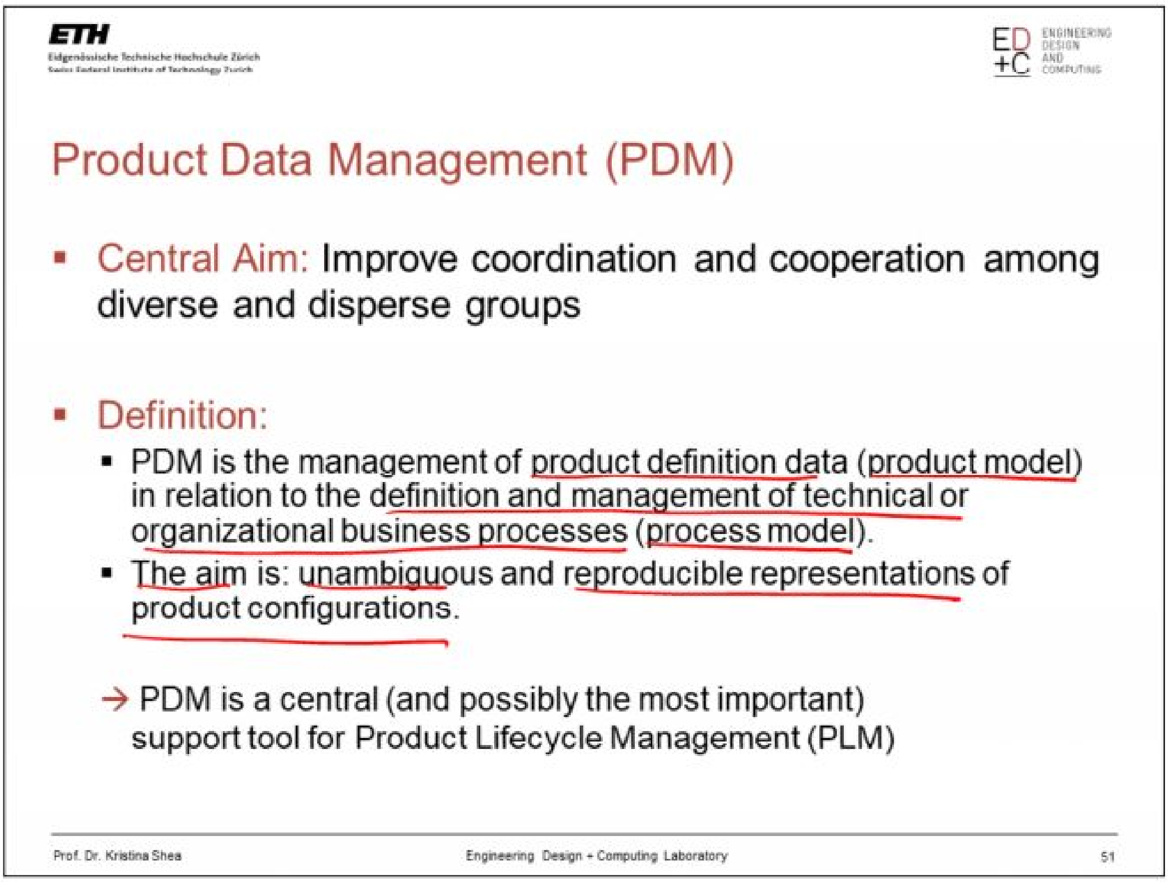
**Other data and information:**

- e.g. owner, company ,copyright

**Specifications of the drawing:**

- Projection method, scale, size, sheet, unit (usually mm)

## Product Data Management (PDM)



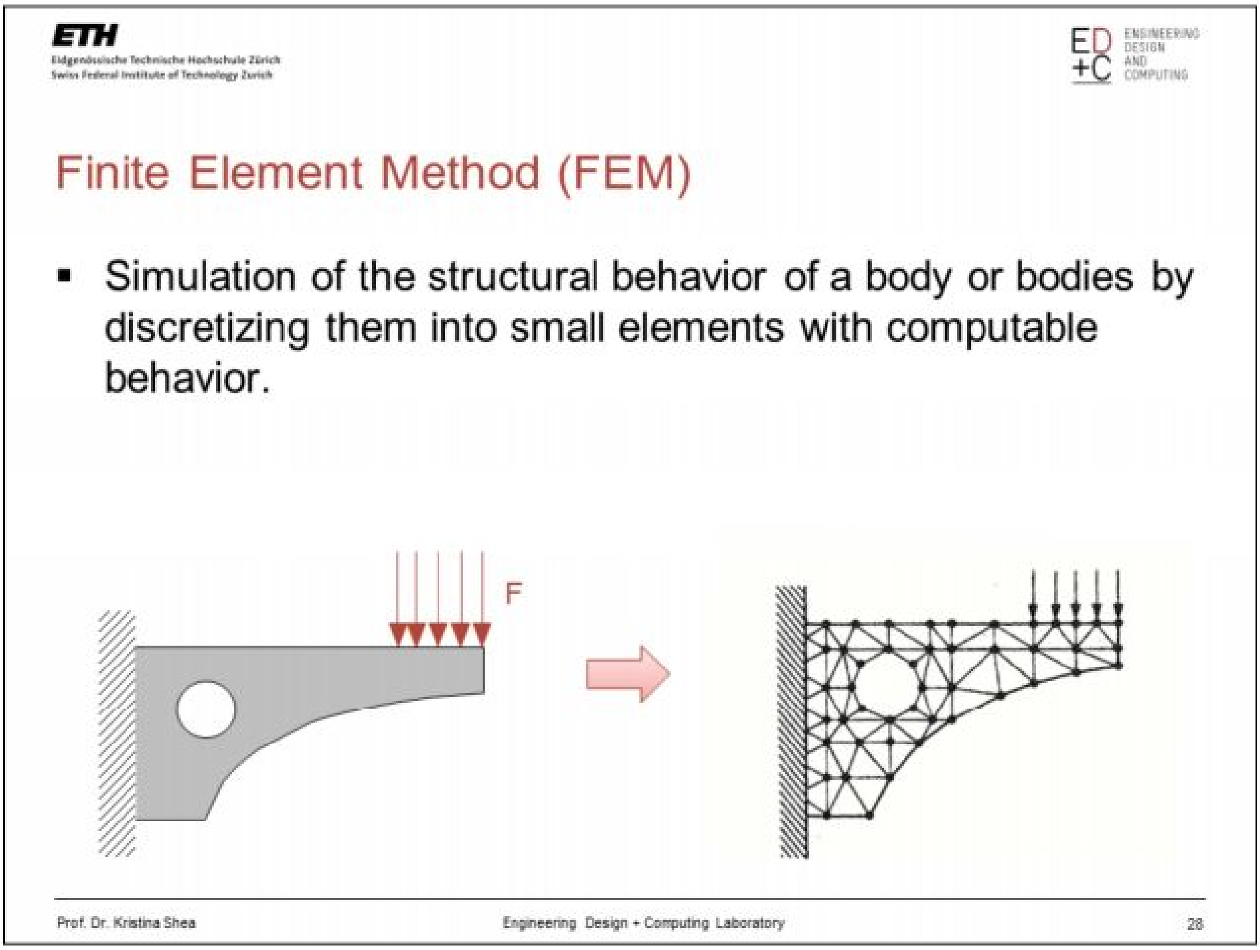

**Product data management (PDM):**

- Structuring (modeling) product information from a wide variety of sources
    - Includes CAD files, project data, notes and documents, test specifications and test analysis reports, cost spreadsheets, numerical control (NC) programs for manufacturing, maintenance procedures, spare part data and logistic support information, etc.
- Central storage of product information (store)
- Central management of product information (manage)
- Accessible product information to all who need it, whenever and where ever (access)
- Integration with other software
- Standardize and support information exchange processes and design change processes
- Standardize and support Engineering Data Management (EDM) and Technical Data Management (TDM) are often used synonymously with PDM.

**Motivation:**

- Globally distributed product development requires processes to become more structured
- Product information is often distributed across different disciplines, company divisions and throughout the entire supply-chain network
- Interactions, communications and decisions, e.g. design changes, traceability
- Everyone needs to have up-to-date information, in real time, anywhere in the world

## Finite Element Method (FEM)




Finite element analysis was originally developed to solve structural problems but now can be used for a wide range of behaviors including heat transfer, vibrations, etc.

Approach:

- Approximate part geometry using a continuum.
- Each body, or part, is represented by mesh(es) of finite elements
- A mesh is composed of elements and nodes
- Solve for the displacements of all nodes (degrees of freedom)
- Each node creates 2n (2D) or 3n (3D) degrees of freedom (DOF)
- Generally (but not without exceptions):
    - Higher number of elements – leads to – Higher accuracy
    - Higher number of elements – leads to – More required computation