Concepts of Pro Engineer Wildfire.

1. Using the Pro Engineer work environment.

2. Using GPS (Geometric Product Specifications) in Wildfire.

2.1. Introduction.

GPS is the complete set of annotations/specifications used on technical drawings to describe the manufacturing quality of the parts.
GPS consists out of 3 parts:
- Dimensional tolerances: limits on dimensions that describe the tolerance one has to machine the part so that it will still function what it is designed for.
- Surface finish: a tolerance which describes the surface finish /smoothness of surfaces.
- Geometrical tolerances which puts limits on form features (straightness, perpendicularity of planes…)

In a parametric CAD-system dimensional tolerances are more than just an extra annotation. In Wildfire tolerances change the real dimension (and thus area, volume and weight) of an object. The object so is a real virtual object, which can be checked in worst case scenarios on dimensional limits and interference/clearance checking.

Surface finish and geometrical tolerances are an integral part of the object, not just annotation on the drawing.

Next to GPS an object in Wildfire also has real material characteristics. Choice of material can be made under EDIT>SETUP>Material.
Any material used in Wildfire has mechanical properties (Young's modulus, coefficient of shear, specific weight) thermal properties (conductivity, heat capacity, coefficient of thermal extension…). These properties are used when calculating material strength with finite elements (Pro Mechanica or Structural and Thermal simulation):
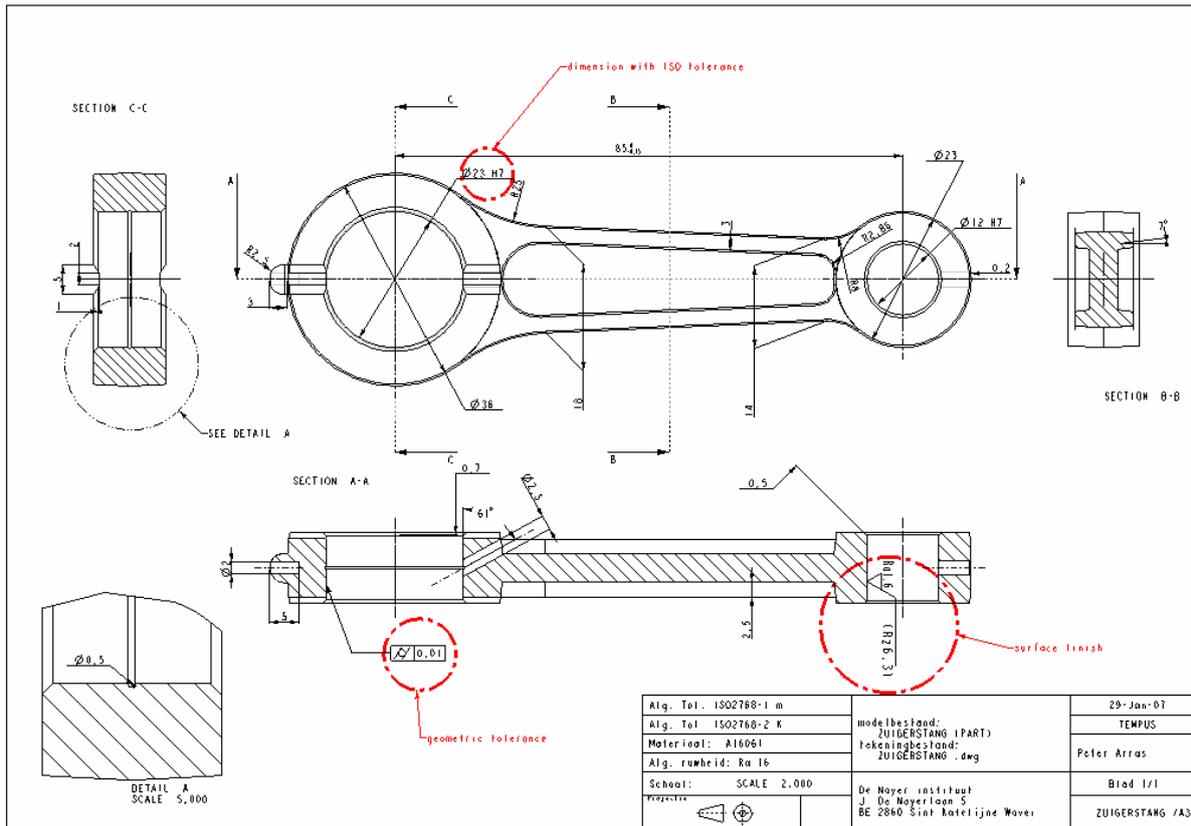APPLICATIONS>MECHANICA.

Figure 1 Technical drawing with GPS annotations

2.2. Dimensional tolerances.

Settings for dimensional tolerances are controlled in 2 ways:
- In part/model mode for the physical aspects of the tolerances. (in EDIT>SETUP>Tol Setup)
- In drawing mode for the visual aspect of tolerances. (in DRAWING PROPERTIES>DRAWING OPTIONS).

2.2.1. *Settings in the model.(EDIT>SETUP>Tol Setup).*

Tolerance standard. (EDIT>SETUP>Tol Setup>Standard)

We make a choice between ANSI (American) or ISO/DIN (International Standard Organisation/Deutsche Industrie Normen).
When designing in mm, the ISO/DIN standard is best suitable.

Model Class.

In ISO/DIN we can pick the model class of tolerances. This describes the easily met tolerances in production, which serves as general tolerance in the model.
In the ISO/DIN system you select one of 4 classes (ISO2768-1): Fine, Medium, Coarse, Very Coarse.
Any dimension in the model which has no specified tolerances, will have the tolerances described in the model class.

Tolerance Tables.

Next to general tolerances, a dimension can have separated tolerances. Ordinary tolerances with limits or plus/minus can be attached easily to a dimension (just right click a dimension, select properties, and change to limits…).
Before using tolerances in the ISO-tolerance symbolic system, you need to load the tables for required classes. (For example holes H, G, shaft g, k..)

2.2.2. _Setting in the working environment (Tools>Options, Tools>Environment)._

The option-file (config.pro) contains all settings for the working environment of Wildfire.
Concerning tolerances it is advisable to put the setting tol_mode = nominal.
This means that any dimension shall standard be a nominal measure, with the general tolerances of the tolerance system. Any other tolerance shall be put on a dimension by means of the dimension properties.
To visualize and use tolerances in model mode, you unblank tolerance in Tools>Environment.

2.2.3. _Setting up tolerances and/or changing tolerance values_

a.   In model-mode

Individual tolerances on specific dimension are set up in dimension properties (select dimension, right click, properties).
-   Tolerance mode: change from nominal to the desired tolerance (limits, plus/minus)
-   Tolerance table: for shafts and holes in the ISO symbolic tolerances system change from general to shaft/hole.
-   Basic dimension: dimension for which the global tolerance isn't evaluated. Tolerance is put up with a Geometric tolerance of position.
-
In the "Dimension text" tab you can put extra text in the dimension: e.g. M before the measured dimension of a metric thread.
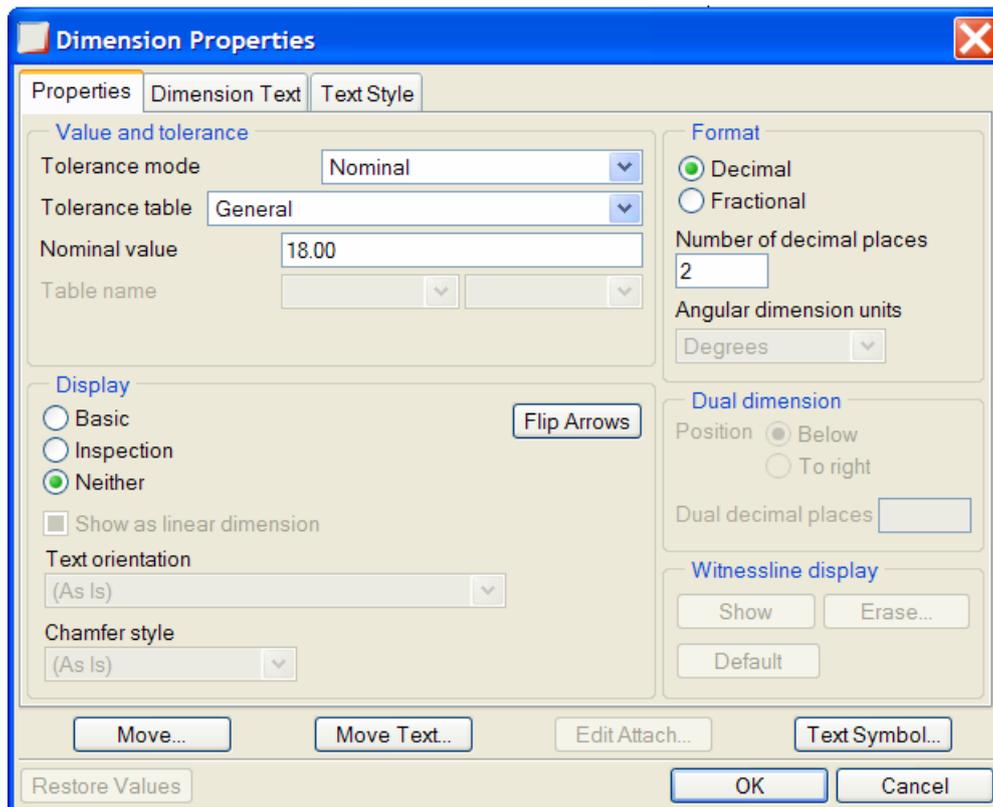
Figure 2 Dimension properties box

        b. In drawing-mode

Tolerances are part of the model. As such they can be shown/erased with the dimensions and axis lines. It is best to put all the dimensional tolerances in the model. (In FILE>PROPERTIES>DRAWING OPTIONS you control the visibility setting of tolerances.) In drawing mode, setting up tolerances to a dimension is done in the same way as in model mode: select dimension, right click and properties setup.
When tolerances are put up in drawing mode, they are automatically assigned to the dimensions in the model.

2.2.4. *Setting dimensions to limits in a part.*

In Edit/setup there is Dim Bound. With dimension bounds, you can set a dimension to its nominal value, to the upper or lower limit value, and to its mid value.
Depending on what you are going to analyse you should set the part according to the worst case.
For example, if you are going to calculate mechanical stresses, the volume of material should be at its lowest possible value: so dimension of the outside of the part should be set to lower limits, and internal dimensions to upper limits.

2.2.5. *Tips and best practices.*

1.   Always design in nominal value first.
2.   Change dimension with specific tolerances according to the required tolerance system and values.
3.   Set up tolerances as much as possible in the model, not in the drawing.

2.3. Surface finish.

Surface finish is setup up in EDIT>SETUP>Surface finish in model/part mode.
Under SETUP you put up the symbol that indicate the surface finish: CREATE.
The definition sequence is as follows:
- Indicate the plane to which the symbol will be parallel (this is the orientation of the 2D symbol in space. The arrow indicates the front of the plane.
- Enter the rotation angle: the angle relative to the arrow in which direction the text will be put up.
- You now enter the dialogue box:
  - Symbol name: choice of "generic", "machined", "unmachined" for simple surface finish symbols or browse in the directory tree to "library symbols>isosurfsym" for extended symbol.
  - References: select the surface the finish will be associated with
  - Placement: select with or without leader, than indicate the point the symbol will touch on the reference surface, and place the symbol using the middle mouse button.
  - Grouping tab: select extra annotation on the symbol
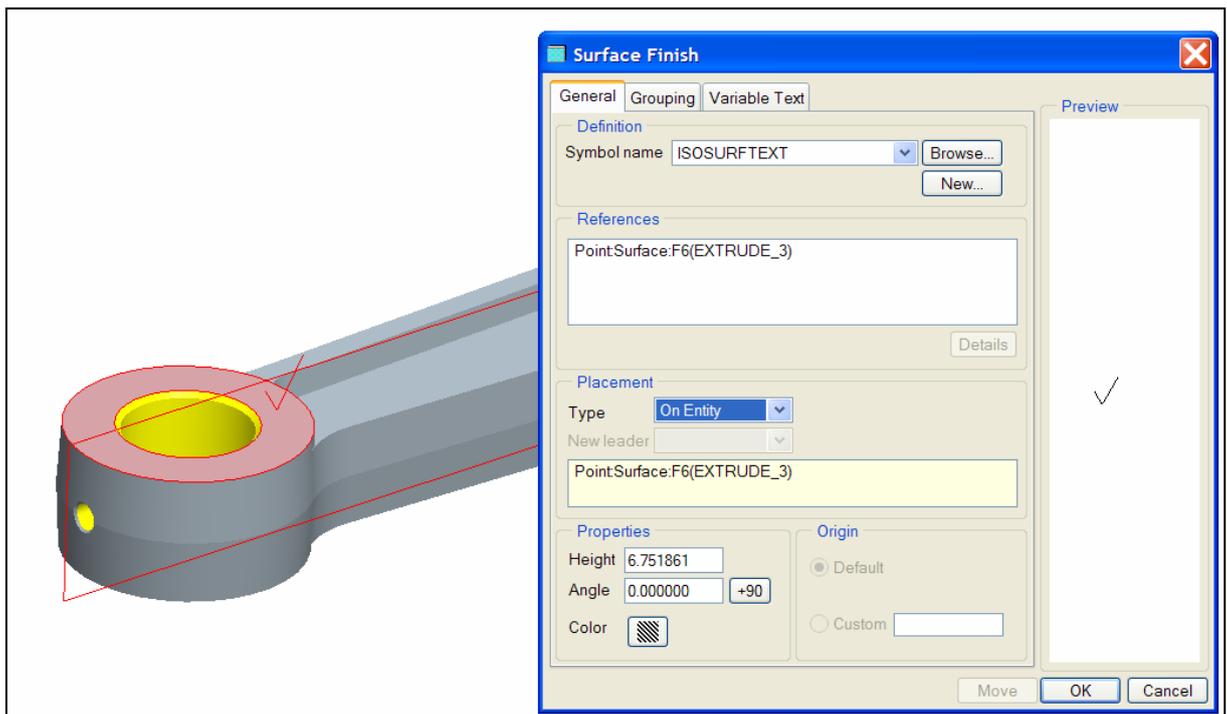
  - Variable text: set up values for the annotations.

Figure 3 Surface finish dialogue box.

In drawing mode you can also setup surface finish symbols: Insert>Surface finish. Since you are in 2D the view plane in which you select the entity to be surface toleranced will also be the plane the symbol will be parallel to.

2.4. Geometric tolerances.

Geometric tolerances describe the tolerance on shapes (marked in green in figure 5), or location of an entity relative to another entity (marked in red in figure 5).
Shape includes: straightness of a line, flatness of a flat surface, roundness of a circle…
Location includes: perpendicular, parallel, axial and total run out, position.

Geometric tolerances are setup in EDIT>SETUP>Geom tol>CREATE in model/part model.
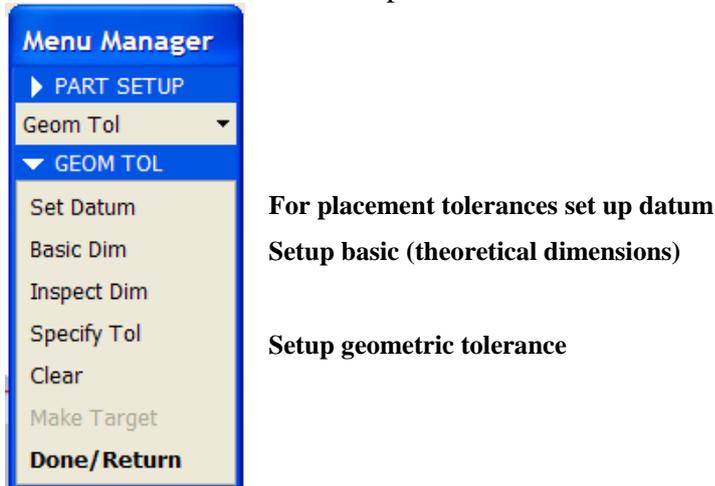
For placement tolerances set up datum

Setup basic (theoretical dimensions)

Setup geometric tolerance

Figure 4 Geom tol menu.

The definition sequence is as follows:
- Setup datum if necessary: a datum can be a datum axis line or a datum plane, depending on the tolerance you are setting. Datum references are necessary when setting up placement Gtols (marked in red in figure 5).
- Specify Tol:
    o Indicate the plane to which the symbol will be parallel (this is the orientation of the 2D symbol in space. The arrow indicates the front of the plane.
    o Enter the rotation angle: the angle relative to the arrow in which direction the text will be put up.
    o You now enter the dialogue box:
        ▪ Select geometric tolerance symbol name at the left.
        ▪ Model Refs:
            • Model: select the model the Gtol will be in
            • Reference: select edge/surface/axis/entity (depending on the required tolerance) the geometric tolerance will be associated with
        ▪ Placement: select with or without leader, than indicate the point the symbol will touch on the reference surface, and place the symbol using the middle mouse button.
        ▪ Datum Refs tab: select datum (plane, axis) with a placement GTol (for Gtols marked in red in figure 5).
        ▪ Tol value: set up value for the annotation (nr 5 in figure 6).
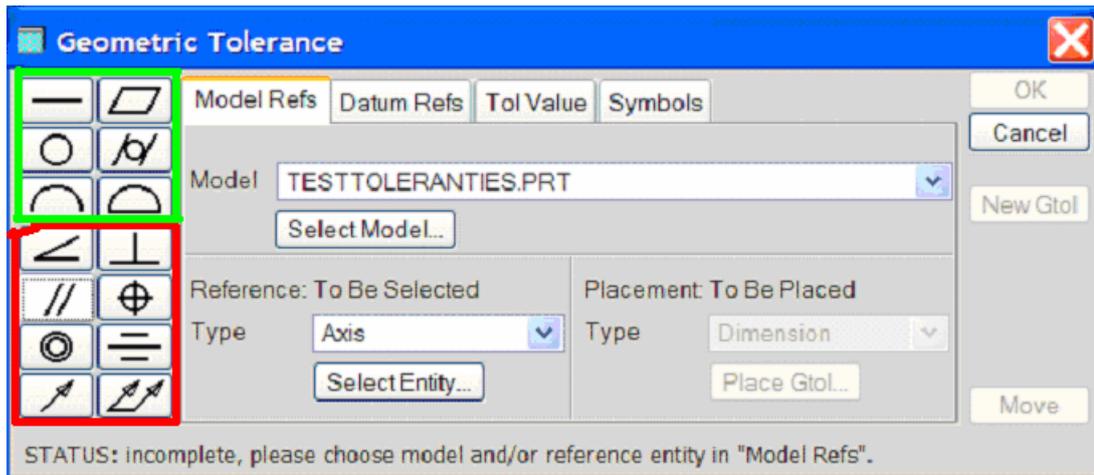   ▪ Symbols: select extra symbols

Figure 5 Geometrical tolerance box

Geometric Tolerance Layout

Pro/ENGINEER specifies a GTOL for an individual feature by means of a feature control frame (a rectangle) divided into compartments containing the GTOL symbol followed by the tolerance value. Where applicable, it also follows the tolerance with a material condition symbol.

If a GTOL is related to a datum, it places the reference datum name in a compartment following the tolerance value. Where applicable, it follows the datum reference letter with a material condition symbol.
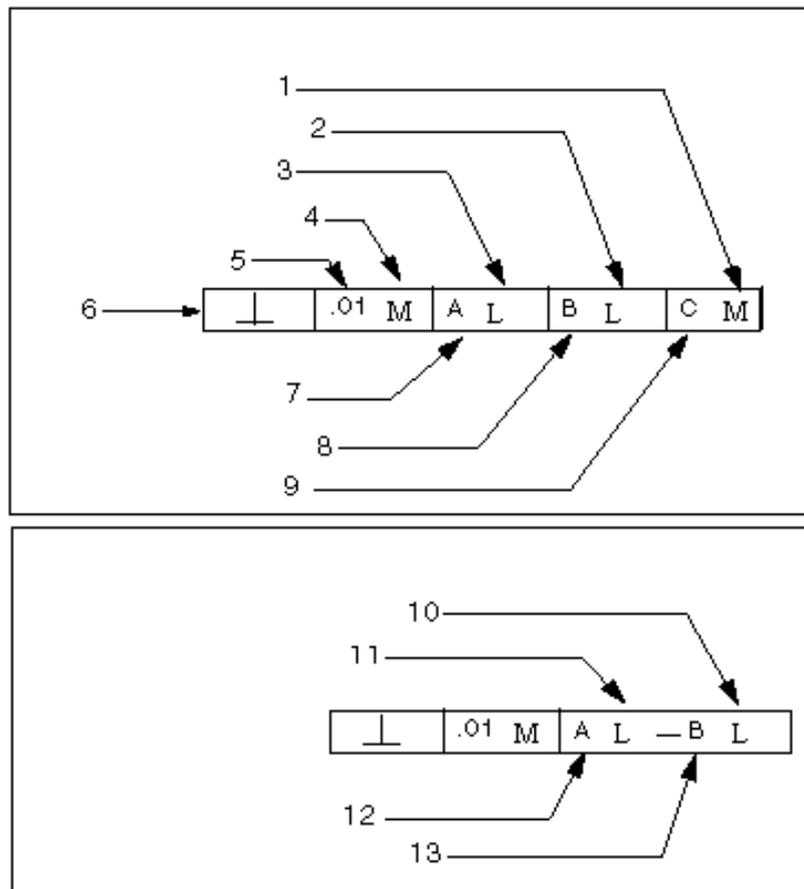

Figure 6 Geometric tolerance layout

1.   Matl Cond, Tertiary
2.   Matl Cond, Secondary
3.   Matl Cond, Primary
4.   Matl Cond, Tolerance
5.   Value
6.   Type
7.   Datum Ref, Primary
8.   Datum Ref, Secondary
9.   Datum Ref, Tertiary
10. Matl Cond, Primary, Compound
11. Matl Cond, Primary, Basic
12. Datum Ref, Primary, Basic
13. Datum Ref, Primary, Compound

Geometric tolerances can also be set from within drawing mode: Insert>Geometric Tolerance.

For an understanding of Geometric Tolerances: see theory on technical drawings.