Working With Tolerances

Most CAD systems have functionality to allow the user to add tolerance information to dimensions. This allows drawings to reflect design or manufacturing intent but since the tolerances are nothing more than an additional text note applied to the dimension no additional functionality can be gained from the tolerance value. These 'unintelligent' tolerances can serve no benefit for analysing such things as tolerance build up problems and volume/weight variations.

Pro Engineer builds three-dimensional models and allows the defining dimensions to have tolerances applied to them. These tolerances can be displayed in the three-dimensional model window as well as appearing on the subsequent drawing derived from the model. Since these tolerances are inherent in the defining dimensions of the part they can be used in a much more 'intelligent' way for analysing the parts performance in an assembly as will be shown in the following scenario.

Tolerance Overview

All Pro Engineer components are built with nominal values. The nominal values represent the ideal sizes that are intended by the designer. However manufacturing practice dictates that no dimension can be guaranteed exactly. All manufacturing processes require a dimensional tolerance range within which the size can be guaranteed. The smaller the tolerance range required the more expensive the manufacturing process required is likely to be. Assigning very narrow tolerances to every dimension causes a component to be more expensive than is necessary to achieve the function intended. It is the designers' role to analyse the product and decide which dimensions are critical to achieving the product function.

All dimensions entered in Pro Engineer are given a default tolerance value. To see these values make sure that the option DIMENSION TOLERANCES is checked in the TOOLS > ENVIRONMENT dialog. Every dimension for a feature will now be displayed with tolerances shown. Also the part window will display the default tolerances as follows…

X.X	+- 0.1
X.XX	+-0.01
X.XXX	+-0.001
ANG	+-0.5

This shows that the default tolerance varies according to the number of decimal places assigned to a dimension. The number of decimal places is determined whilst in the sketcher by SKETCH > OPTIONS on the PARAMETERS tab under NUM DIGITS. The default values, shown above, can also be changed choosing ANNOTATION in the selection filter at the bottom of the screen and double clicking on the tolerance value. Any modifications you make to these default tolerances apply only to dimensions subsequently created. Previous dimensions will have the default tolerances active when they were created. If, for example, you normally work with a general tolerance of +/- 0.3 on all unspecified dimensions, you could change all of the linear values to 0.3 and then assign suitable tolerances to individual dimensions as required later.

Assigning Tolerances To Part Dimensions

The two components shown in Figure 1 and Figure 2 are simple parts which are intended to fit together as part of an assembly. They are to be constructed as two separate Pro Engineer parts called tol1 and tol2. Using your existing knowledge of Pro Engineer create these two separate parts now using mmns_part_solid template. The L shaped block in tol1 can be created as an extruded protrusion and the three pegs are a second extruded protrusion created by sketching three circles onto the top face. **ENSURE YOU DIMENSION THE PARTS EXACTLY AS SHOWN.**

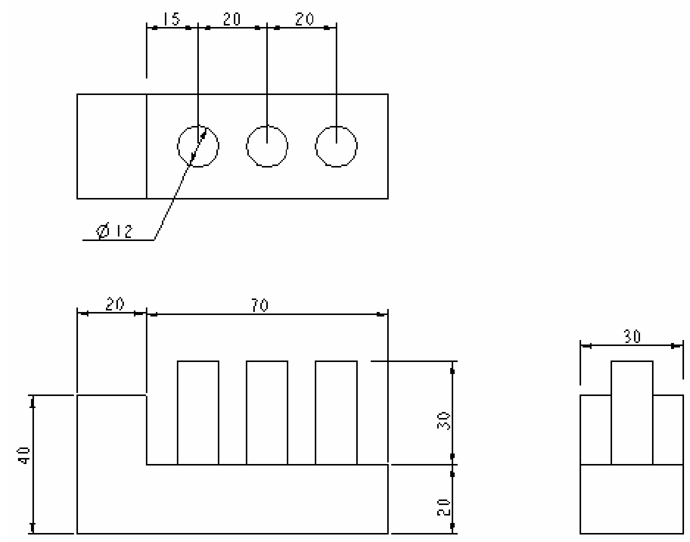

Figure 1 : An Engineering Component Called *tol1*

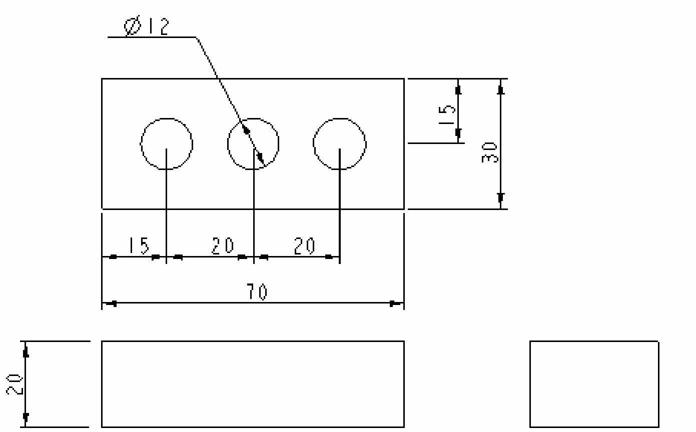

Figure 2 : A Second Engineering Component Called *tol2*

Having created the two parts it would be interesting to see what tolerance has been applied to each dimension. Open part tol1 now. First make sure that the DIMENSION TOLERANCES is checked in TOOLS > ENVIRONMENT. The default number of decimal places in sketcher is 2. If you have not changed this or the default tolerance values the tolerance on all dimensions with two decimal places X.XX should be reported in the graphics window as +/- 0.01. Confirm the tolerances are as you expected by right clicking on a feature in the model tree on the left of the screen and choosing EDIT. The dimensions should now be shown with their upper and lower limit values as you can see in Figure 3.

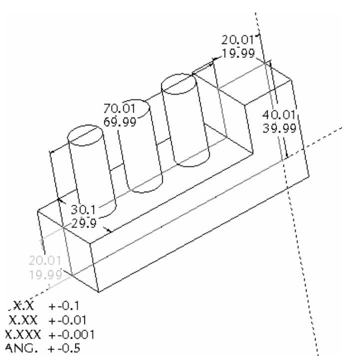

Figure 3 : Tolerance Shown

As an example of what tolerances can be used for at a part level the volume of material in a component could be calculated. Clearly since the dimensions have a tolerance the volume would also have a tolerance. To find out the volume range we need to calculate mass properties at maximum and minimum material conditions. The part dimensions are set to an extreme value by EDIT > SETUP > DIM BOUND > SET ALL > UPPER > DONE. Notice that SET ALL can be used for tol1 since maximum material conditions are when all dimensions are at the upper limit. If the part had a hole the dimensions for the hole feature would need to be set to the lower limit for maximum material. The volume can now be calculated using ANALYSIS > MODEL ANALYSIS and choosing the Type as Model Mass Properties and Compute. The density can be set to 1. Note the volume is calculated as 76 495 mm^3. The calculation can now be repeated after first setting the dimensions to minimum material condition using EDIT > SETUP > DIM BOUND > SET ALL > LOWER > DONE. Note the volume is calculated as 75 863 mm^3 a total variation of 632 mm^3. Use EDIT > SETUP > DIM BOUND > SET ALL > NOMINAL > DONE to reset the bounds to their normal state.

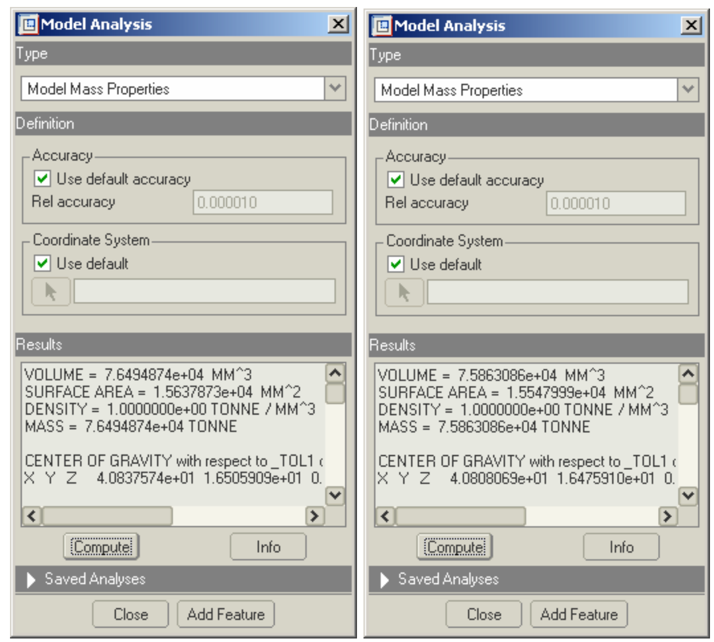

Figure 4 : Mass Property Calculations

Showing Tolerances on Drawings

To show dimensions on a drawing is a simple matter so long as the system has been set-up correctly. The following paragraph describes how to set up the system. You should be working in a drawing for these commands to work. Refer to Tutorials 4 and 5 for creating drawings and adding dimensions. Create a drawing of part tol1 now positioning the dimensions as shown in Figure 5

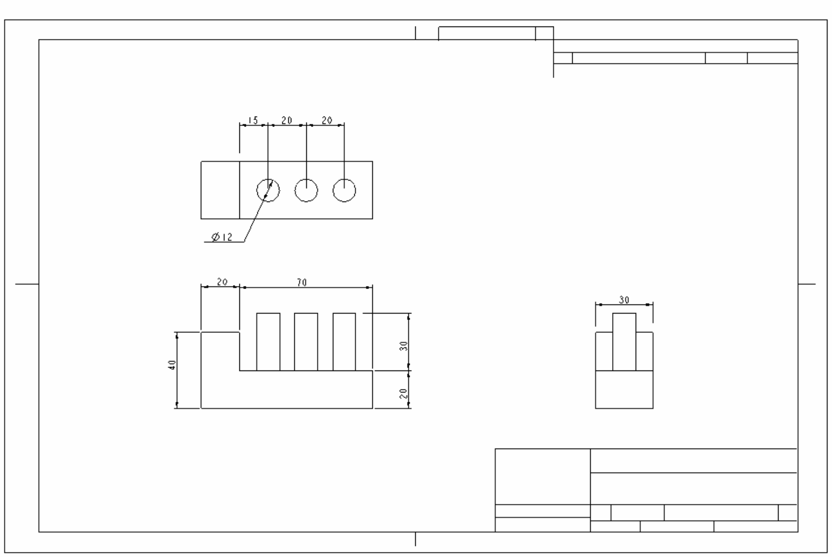

Figure 5 : Initial Drawing

The configuration for your drawing appearance (text height, arrow size etc.) is stored in a drawing set-up file. The current settings can be altered using FILE > PROPERTIES >DRAWING OPTIONS. A new dialog window will appear showing all your current settings. If tolerances are to appear in your drawing is important that this file contains a line that says tol_display yes. In the Option field type tol_display and set the value field to yes. Click Add/Change to change the value then close the dialog.

Now that the drawing is set correctly any dimension(s) can be selected and the command EDIT ⇒ PROPERTIES given. A form will appear showing the dimension settings as shown in Figure 6. The Tolerance Mode should be set to NOMINAL if you do not want tolerances displayed.

Choose LIMITS or PLUS-MINUS to display tolerances. Notice that you can also adjust the value of the dimension and the tolerance from here.

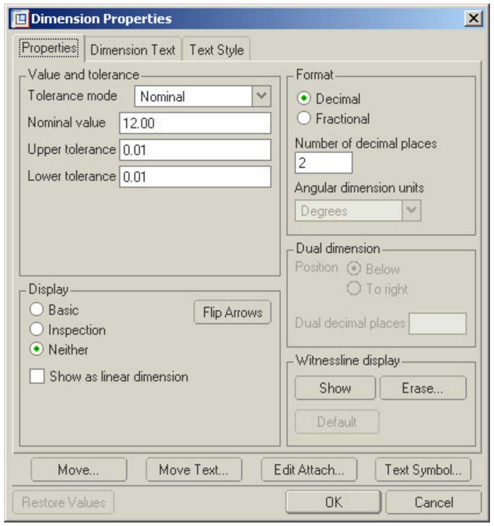

Figure 6 : The Modify Dimension Form

Select the 12 diameter dimension of the pins and change the Tolerance Mode to Limits now to see the difference.

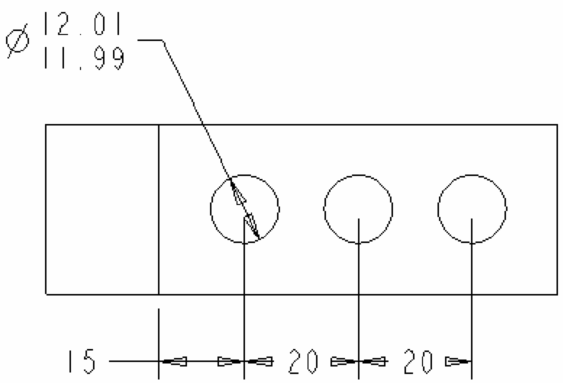

Figure 7 : A Dimension with Tolerance

Analysing Assemblies with Tolerances

The holes and pins in parts tol1 and tol2 are intended to assemble together with a clearance fit of H9e9 as designated by BS4500. Reference to this standard shows that the tolerance for a 12mm shaft of this specification is –0.032 to –0.075. The matching hole would be +0.000 to +0.043. Change the tolerance of these two features now. In the tol1 part right click on the feature for the pins in the model tree and choose EDIT. Since you just set the tolerance mode for the diameter dimension to Limits two diameter values (12.01 and 11.99) should be displayed. Double click on the lower diameter dimension value for the pins (11.99) and type in the new value of 11.880. Double Click on the upper dimension value (12.01) and type in the new value of 11.950. In tol2 you will need to Edit the feature then click on the diameter dimension and choose EDIT > PROPERTIES so that you can set this to Tolerance Mode Limits. Then repeat the changes setting the lower dimension to 12.000 and the upper to 12.043.

To illustrate these concepts create a new empty assembly called tolass and assemble the two parts, tol1 and tol2, together. Use tol1 as the first component. Apply three constraints as tol2 is placed as shown in Figure 8. Make sure that the first mate constraint references the end from which the 15 dimensions is taken on tol2.

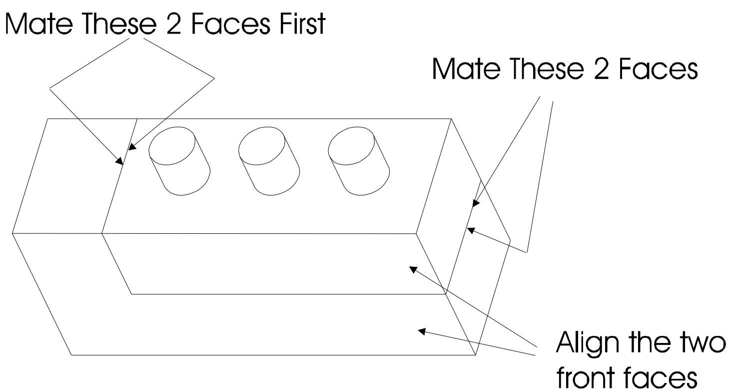

Figure 8 : The Assembly Constraints

Once the assembly is complete analysis can take place. One problem that often occurs with tolerances is that assembled components will not work correctly when the parts in an assembly are all at one extreme of size. ProEngineer can calculate whether two parts in an assembly interfere with each other. In the assembly tolass issue the command ANALYSIS > MODEL ANALYSIS and choose the type of analysis as PAIRS CLEARANCE. Pick the two parts in the assembly and Compute. At this stage ProEngineer should report a zero clearance. Of course since the two parts are designed to touch along two faces this is what you would expect. What is the clearance between the pins and their holes? To find this use the command ANALYSIS > MODEL ANALYSIS and choose the type of analysis as PAIRS CLEARANCE but this time choose SURFACE as the From and To option. Pick on the cylindrical surface of one pin then right click until you pick the cylindrical surface of the corresponding hole. ProEngineer should report a clearance of 0. 0.0529799mm and a red marker will be displayed at one of the points of minimum clearance. Why is there this clearance value? Because the calculations are currently being performed on the parts at nominal sizes. The calculation is…

Pins at (11.88+11.95)/2=11.9150mm diameter.

Holes at (12.043+12.000)/2=12.0215mm diameter.

Clearance of (12.0215-11.915)/2=0.053mm.

If you do not get this value make sure you have set the dimension bounds to nominal (EDIT > SETUP > DIM BOUND > SET ALL > NOMINAL > pick part > DONE for each part).

The question remains that if the parts are at their worst extremes of tolerance will the parts still assemble? What are the worst extremes? This clearly occurs when the pins are at there biggest and the holes are at there smallest. Also if the distance between the pins is at a minimum and the distance between the holes is a maximum (or vice versa) a worst case scenario exists. To achieve this condition in ProEngineer we need to set up the dimension bounds as we did before but in the assembly. Use EDIT > SETUP > DIM BOUND > SET SELECTED > LOWER and pick the part tol1 on one of the pins. The dimensions for this feature will be displayed and you can now pick each spacing dimension in turn (i.e. 15mm, 20mm and 20mm) followed by DONE. These will be displayed in white indicating they are set to lower. Now using the similar command set the diameter of the pins to the UPPER tolerance. This time the dimension will be displayed in grey indicating an upper tolerance. Repeat this procedure for the holes in tol2 (use right click until you pick on the holes). This time the spacing needs to be set to UPPER and hole diameters to LOWER.

The assembly is now set to one extreme of tolerance. If the interference analysis between the two parts is performed again interference will be reported the red areas where the parts overlap is highlighted. The assembly will NOT work as intended with the current tolerances. To make it work the tolerances on the spacing of the holes could be reduced but this is likely to increase the cost of the component. Is there an alternative?

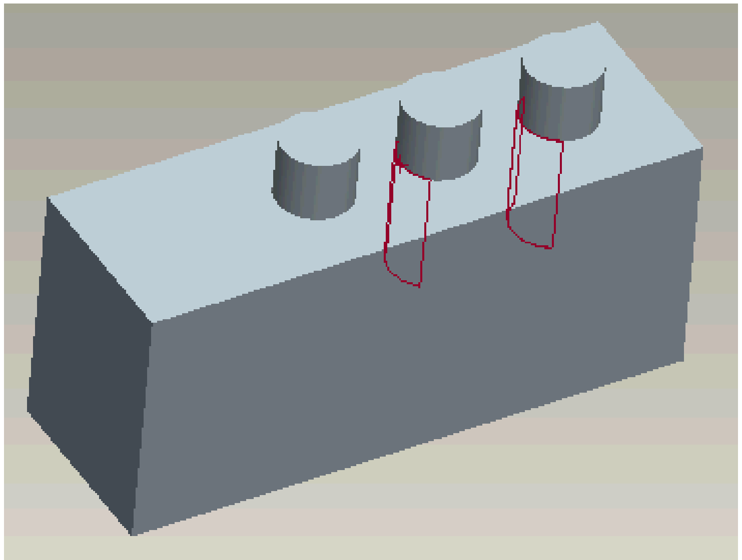

Figure 9 : Interference

If you look at the previous interference analysis you can see that the interference occurs on the last two pins, why? The problem is called tolerance stack up. Consider Figure 10 which shows two alternative dimensioning schemes.

The two schemes apparently are no different until you consider tolerances. If a tolerance of +/- 0.01 is applied to each dimension what is the overall distance to the centre of the right most circle. Using the chain-dimensioning scheme the tolerances are cumulative so the answer is 55 +/- 0.03. Using the baseline dimensioning scheme the dimension is specifically stated so the tolerances do not add up and the answer is 55 +/- 0.01. An improvement in accuracy has been achieved with no tightening of tolerances and no extra cost. In general baseline dimensioning is more accurate and should always be used except when chain dimensioning better reflects the critical dimensions.

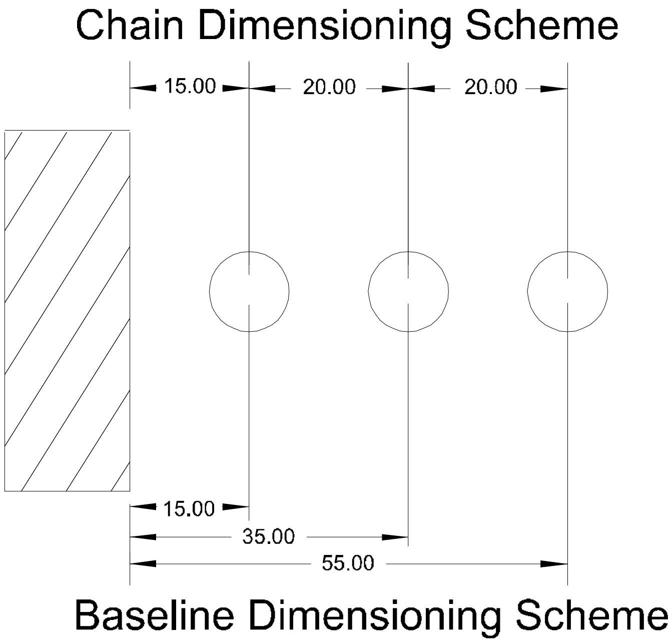

Figure 10 : Alternative Dimensioning Schemes

You may like to return to the parts tol1 and tol2 and EDIT DEFINITION on the features so that the dimensions for the holes and pins reflect the baseline scheme. If you perform the analysis again you will find there is no interference and the assembly will work as intended.

Review

So what should you have learnt?

- How to define tolerances on part dimensions.
- How to show tolerances on drawings.
- How to use tolerances for analysis.

Any problems with these? Then you should go back through the tutorial – perhaps several times – until you can complete it without any help.