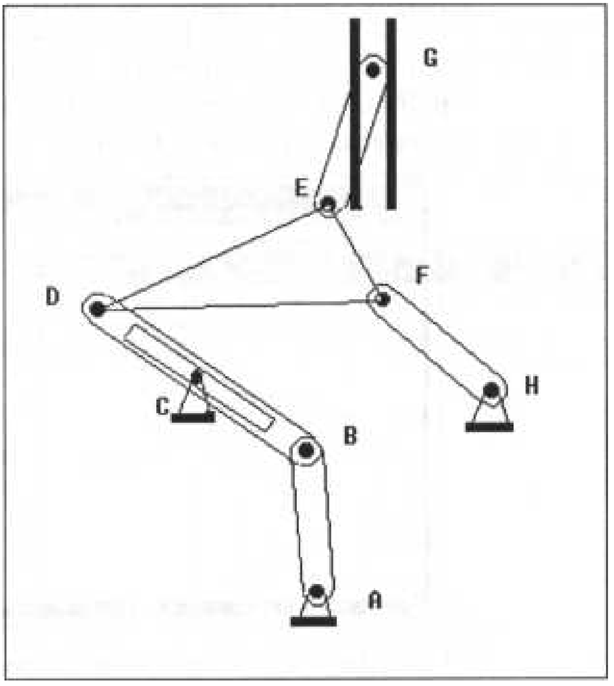
Analytix

design the future

Worked Examples

Example 1: A Piston Mechanism



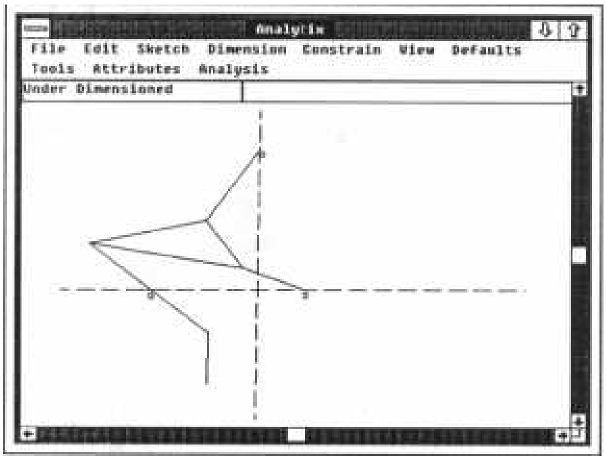
The Problem

The above is a schematic diagram of a variable-stroke engine mechanism. If crank AB rotates at 2000 rpm, give displacement, velocity and acceleration profiles for the piston G.

Fixed points C and H are on a horizontal line, and point G is constrained to slide in a vertical slot. The perpendicular distance from this vertical line to point H is 6. The following are various distanced measured from the figure: AC = 12; CH = 18; AH = 17; AB=7; BID =20; DE=19; EF=7; DF=22; EG= 14; FH = 9.

The Model

We first draw a vertical construction line for piston G to lie on and a horizontal construction line for points C and H. We draw lines representing AB, BD, EG and FH, and we draw triangle DEF. We finally draw point C on the intersection of the horizontal construction line and the line DB.



We now need to dimension the mechanism then specify which piece of the mechanism is to stay fixed.

1 - Add line length dimensions to specify the lengths of lines AB, BD, DE, EF, DF, EG, FH.

2 - Add Distance Point to Point dimensions between C and H, A and C, A and H.

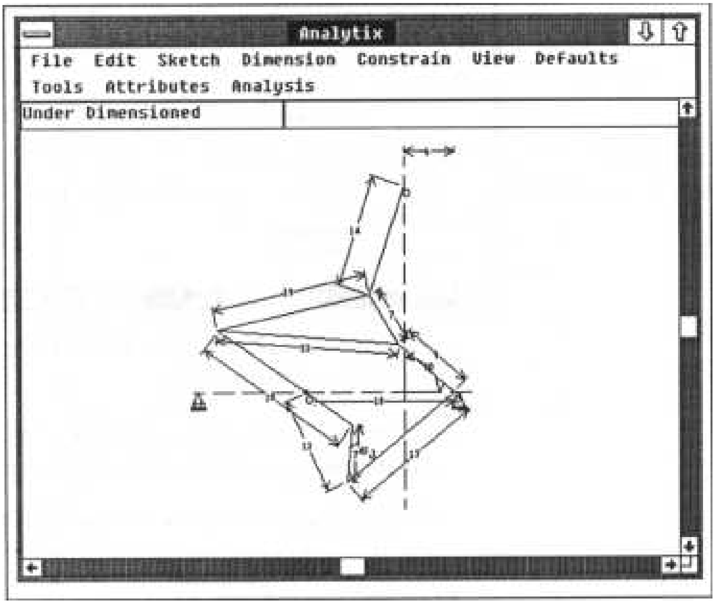
3 - Add Distance Line To Point dimension between the vertical construction line and point H (distance = 6).

4 - Add an angle between the two construction lines (90 degrees).

All the dimensions we have specified so far will stay constant during the motion of the mechanism. It remains to add a driving dimension. This will be the angle of crank AB. In order to give a convenient baseline for this angle, we sketch in the line AH .

Now we add the angle BAH. (any angle will do as we have to study the whole cycle).

To set the fixed elements of the drawing, select point H and the horizontal construction line, then choose Constrain/Fix Point/Line.

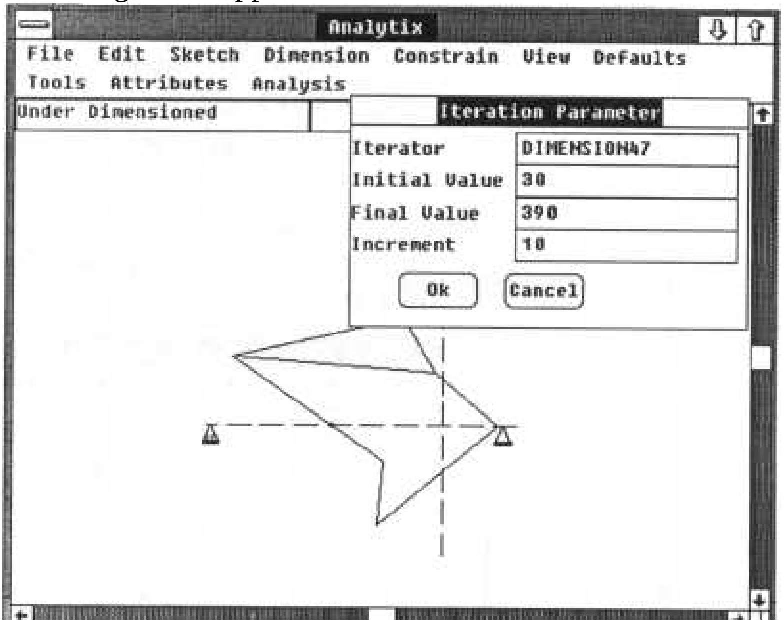


The Solution

First we can animate the model to make sure it actually does what it is supposed to.

Choose Animate *from the Tools menu.*

You will see the iteration box. You should make sure the text cursor is in the edit control labelled Iterator, then go back to the drawing and click on the angle dimension which drives shaft AB. The name of the angle will appear in the Iterator edit control.



Fill in the Initial Value, Final Value, and Step Size to make a complete cycle.

Before starting the animation, you should blank the dimensions, so the drawing is not so cluttered.

Do this by choosing ***Select all Dimensions*** *from the Edit menu them choosing Blank from the View menu.*

Click on OK in the Iteration Box to start the Animation.

Setting the velocity

We are now in a position to derive the displacement of point G at any point in the cycle. However, in order to derive its velocity and acceleration, we need to enter the angular velocity of the crank. The steps are as follows:

1 - Set the default units for angular velocity to be revolutions. (Use Defaults/System Defaults.)

2 - Unblank the dimensions. (Use View / Unblank all.)

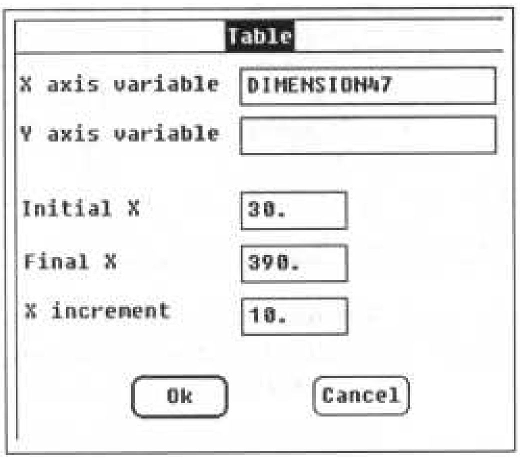
3 - Select the driving angle, then choose Info from the Attributes menu.

4 - Set the velocity to 2000.

Deriving results

We wish to look at the functions ycoord(), yvel(), and yacc() as applied to point G, over the range of motion of the mechanism.

Use Table or Graph from the Tools menu to give you either tables or graphs of the required results.



The values entered for the animation should already be in place for the independent variable, the initial and final values and the step size. You need only add the dependent variable.

In the Y axis variable box type ycoord(

In the main diagram select the point corresponding to G

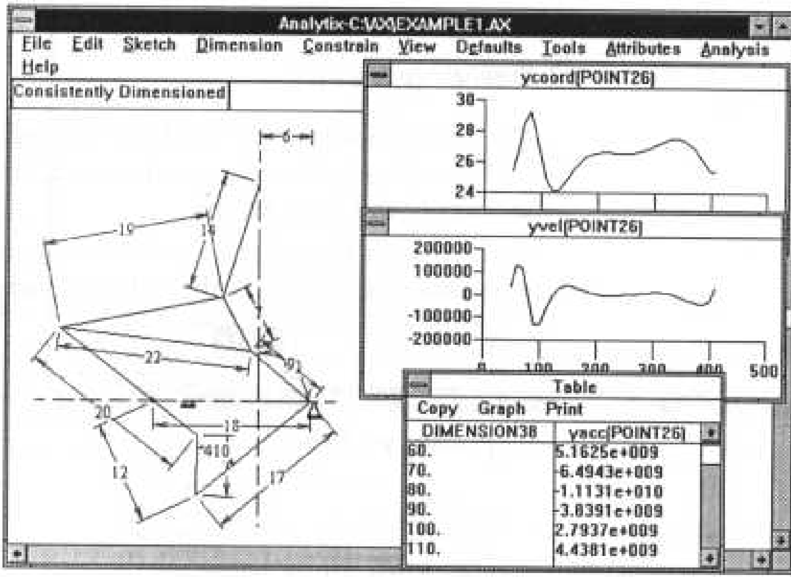
The edit control should now contain:

ycoord(POINT37

(Although the point number will probably be different in your drawing).

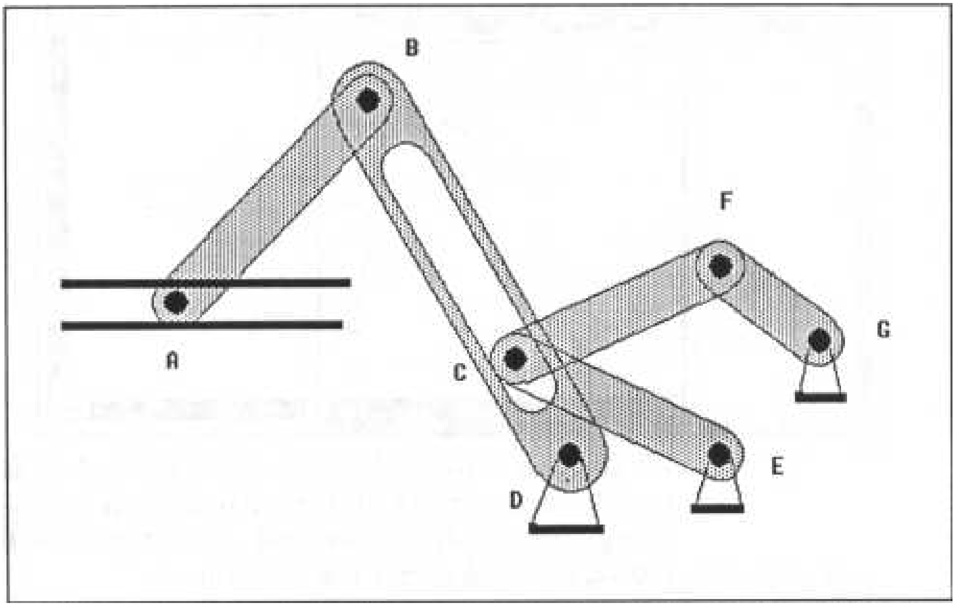
Complete the entry by closing the parenthesis.

Press Ok to create the table or graph.



A similar sequence of commands will let you construct graphs or tables of yvel(POINT37) and yacc(POINT37).

Example 2: Piston Mechanism 2



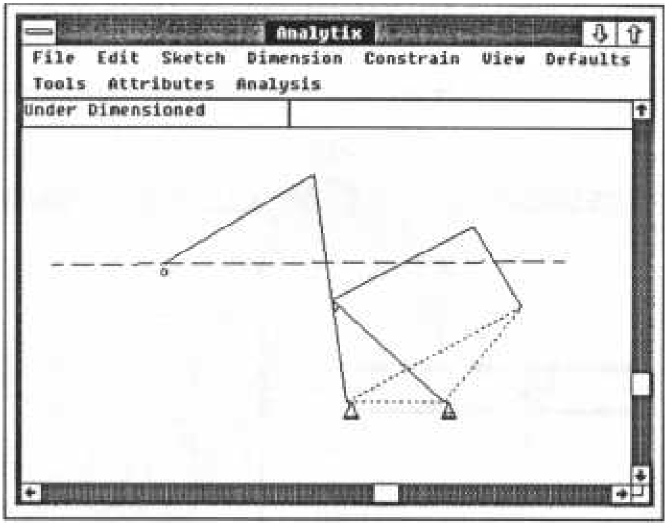
The Problem

Over a cycle of the mechanism, find the torque output on arm FG as a result of a unit force applied to piston A.

Find the force in arm CE.

D and E are on the same horizontal line. A is constrained to lie in a horizontal cylinder distance 6 above D. AB =11; BD=16; EC=9; CF =9; FG=5; DE =6; EG=6; DG =11.

The Model



We sketch a horizontal construction line for A to lie on, a triangle to mark the three fixed points (we change the line style of this triangle to emphasize the fact that it is not part of the mechanism.

Lines are added for each bar of the mechanism, and we fix point D and the line DE.

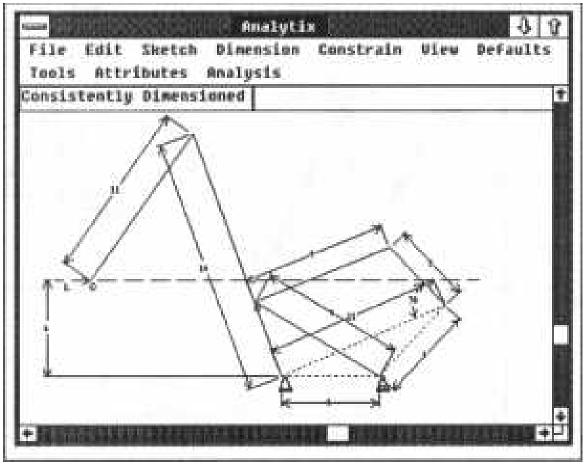
Dimensions are added as follows:

1 - Parallel distance between the construction line and DE.

2 - Lengths of lines AB, BD, CE, CF, FG, DE, DG, EG.

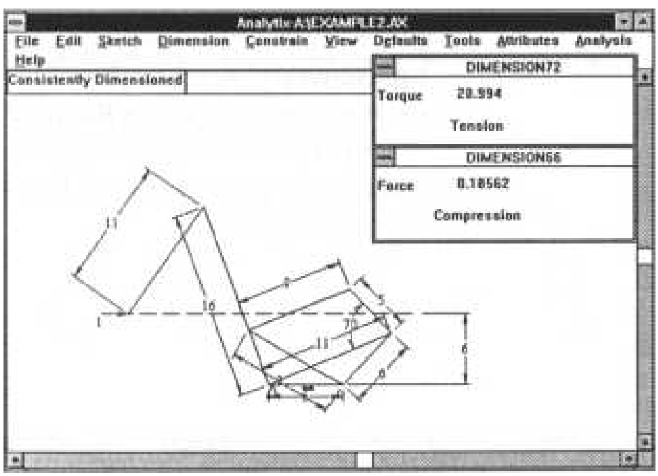
All these dimensions will stay constant during the motion of the mechanism. We now need to add the driving dimension. This could be an angle specifying the orientation of crank FG, or a distance giving the position of piston A. Our problem asks for an output torque on FG as a result of a force applied to A. Analytix allows us to ask for resultant torque in an angular dimension, therefore we should dimension the angle of FG with (say) DG.

We now need to apply a unit force to A. We do this by selecting point A then using the Analysis/Add Load menu option.

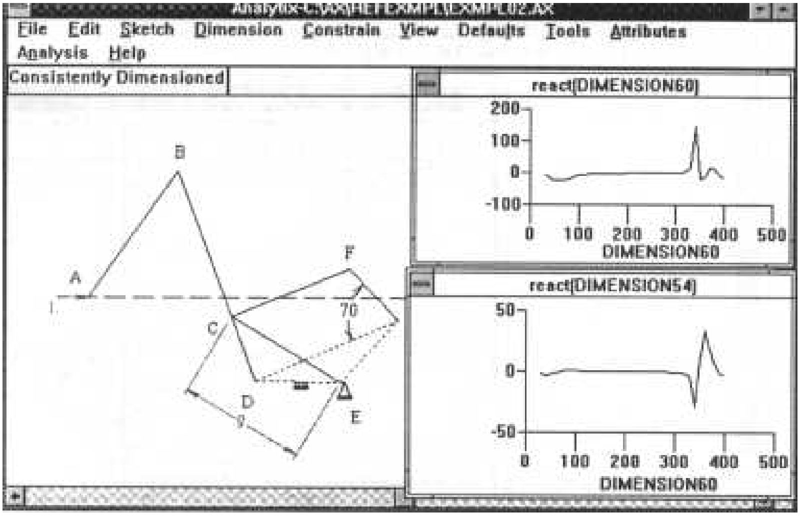


The Solution

To find the torque on FG, we need to find the resultant torque in angle FGD. To do this select the



angle then choose the **Analysis/Resultant Force/Torque menu** option.



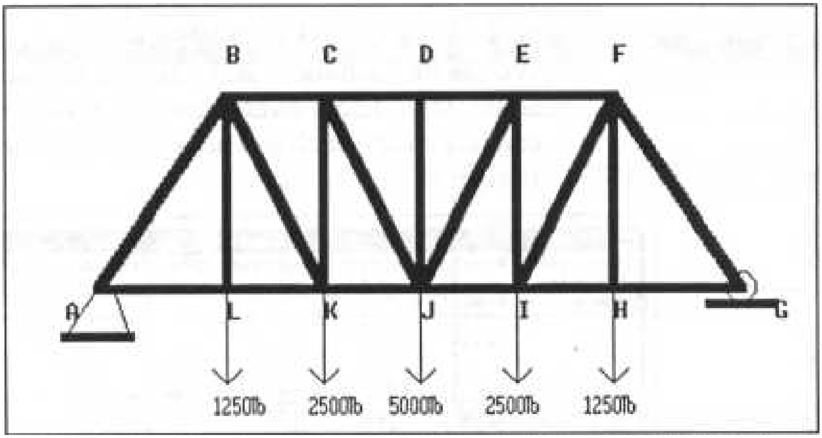
To find the force on bar CE, select the length dimension for that line, then **choose the Analysis/Resultant Force/Torque menu option.**

Now change the angle either by using the Increment tool or using Attributes/Info. As you change the angle you can watch the resultant torque and force alter.

Alternatively, you can produce a graph or table of the reaction force using the react() function with the appropriate dimension name as an argument.

The resulting graphs would perhaps lead us to redesign the linkage.

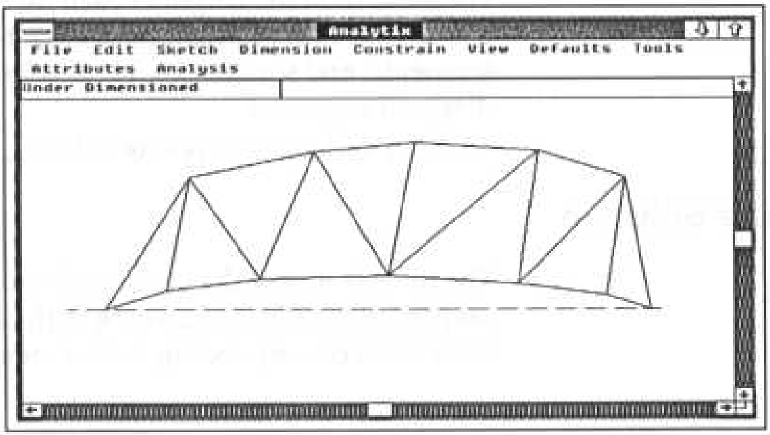
Example 3: Statics of a Bridge Truss



The Problem

In the bridge truss shown, each horizontal member is loft long, and the structure is 12ft high.

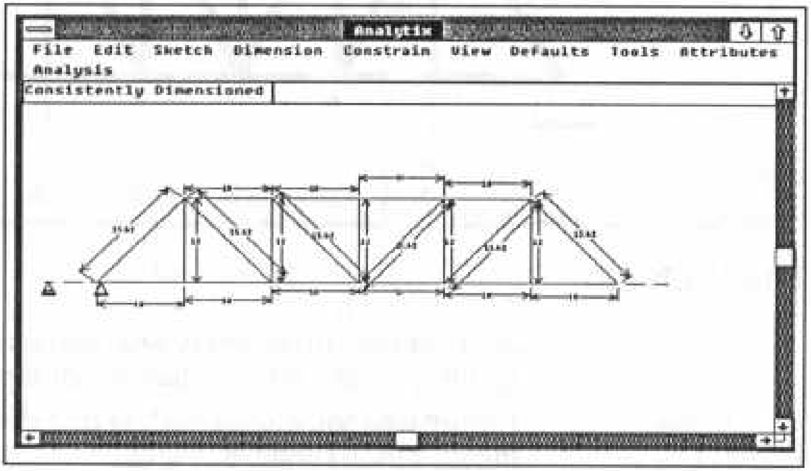
For the load shown, we wish to find the force in member JK, CJ and CK.



The Model

We sketch a construction line to act as the horizontal ground on which the bridge will rest. This construction line will be the fixed line. The point A of the bridge will be the fixed point.

We sketch each individual horizontal member separately. For convenience while sketching (so we can tell where each member ends, we make the horizontal lines arch.)



We dimension the length of each line segment: these lengths are 10 for horizontal segments, 12 for vertical segments, and sqrt(10^2 + 12^2) = 15.62 for the diagonal segments.

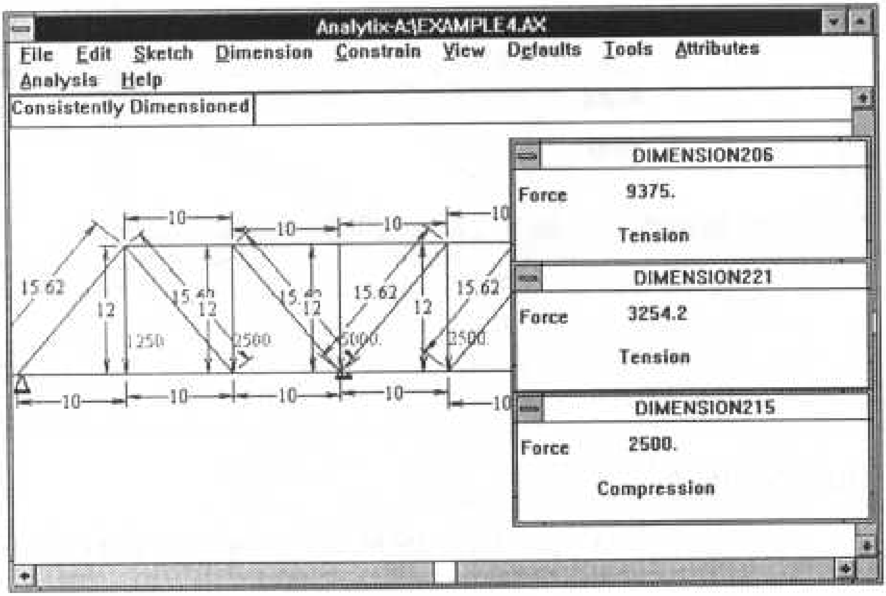
We now add loads to points H,I,J,K,L.

The Solution

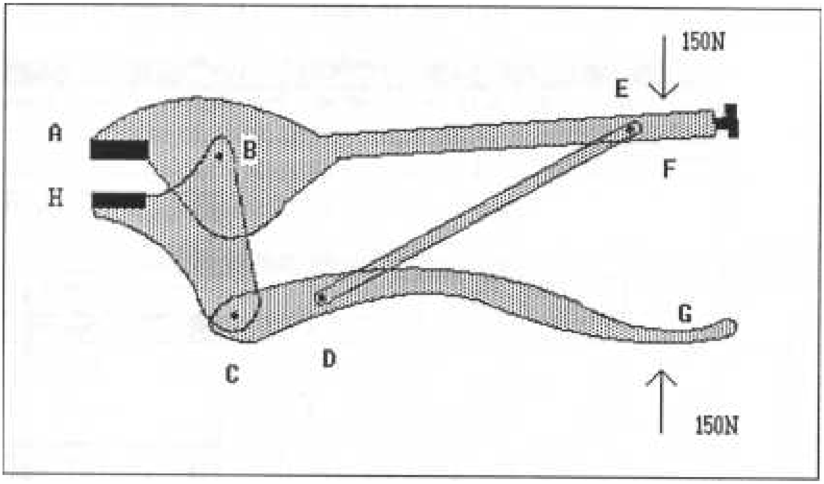
The force in any of the members is obtained by asking for the Resultant Force in the length dimension corresponding to that member.

For each of the members we are interested in, we select the dimension, then select Analysis/Resultant Force/Torque.

Forces in JK, CJ and CK are shown on our screen with JK at the top and CK at the bottom.



Example 4: Vice Grips



The Problem

Opposing 150N forces are applied at points F and G of the pictured handgrips.

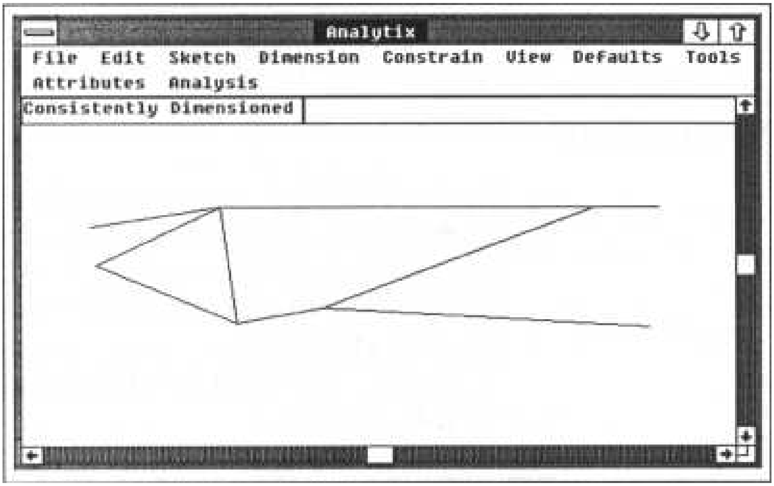
We are required to find the gripping force between A and H when they are 10mm apart.

We are also required to investigate the effect of changing length BE (by adjusting the screw at F.)

Length AB is 30mm, BH is 32mm, BC is 30mm, HC is 35mm, CD is 20mm, DG is 75mm, DE is 67mm, BE is 85mm, and BF is 100mm. Angle ABF is 170 degrees, and angle CDG is 165 degrees.

The Model

We sketch the figure using straight lines to join the critical points as shown.



We dimension the line lengths and angles given. The rule for statics analysis is that all the dimensions should have some corresponding physical entity which acts to preserve the dimension when some force is applied. Hence it is appropriate to specify the angle CDG as this is part of a single piece of metal and thus is being physically constrained. The angle CDE, however, is the angle between two different members which are joined at a pin. There is no physical constraint directly preserving the angle - it is kept constant only as a result of length constraints on other members. Thus it would not be appropriate to enter angle CDE as a dimension in the specification of the geometry.

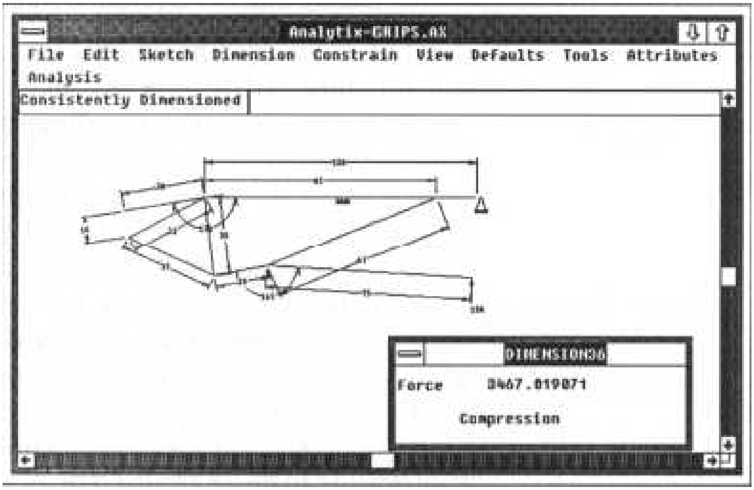
The final dimension we enter is the distance between points A and H, 10mm. Although there is no physical piece of the grips corresponding to this dimension, this is where the reaction force which we wish to measure occurs.

We enter a force of 150 vertically upwards on G. We could also enter a force of 150N downward at F. Instead, however, we fix the point at F. This means that Analytix will automatically add an equal and opposite force to the fixed point F.

The Solution

Having added our externally applied forces to the diagram, it remains to measure the force output by the grippers between A and H. We do this using the Analysis/Resultant Force/Torque menu option.

First we select the 10mm dimension between A and H. Then we pick Analysis/Resultant Force/Torque from the menu.



The Force box appears, showing that this dimension is under compression of 3467N.

To investigate the effect of altering the length of *BE,* we create a graph of this reaction against *BE.*

We do this as follows:

1 - Pick the Tools/Graph menu option

2 - Put the text cursor in the "Parameter t" variable box

3 - Return to the main window and select the dimension between B and E.

4 - Put the cursor in the y-axis variable box.

5 - Type react(

6 - Add the appropriate dimension name by selecting the dimension between the two teeth of the gripper.

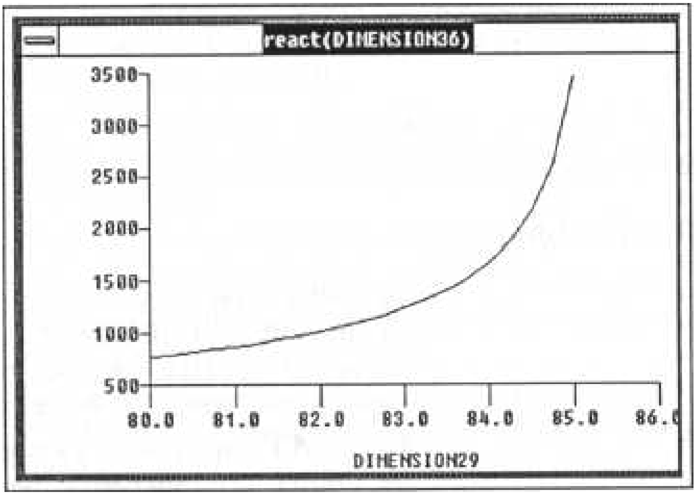
7 - Finish off the expression by closing the parentheses.

8 - Add an initial value of 80.

9 - Add a final value of 85

10 - Specify a step size of 0.25

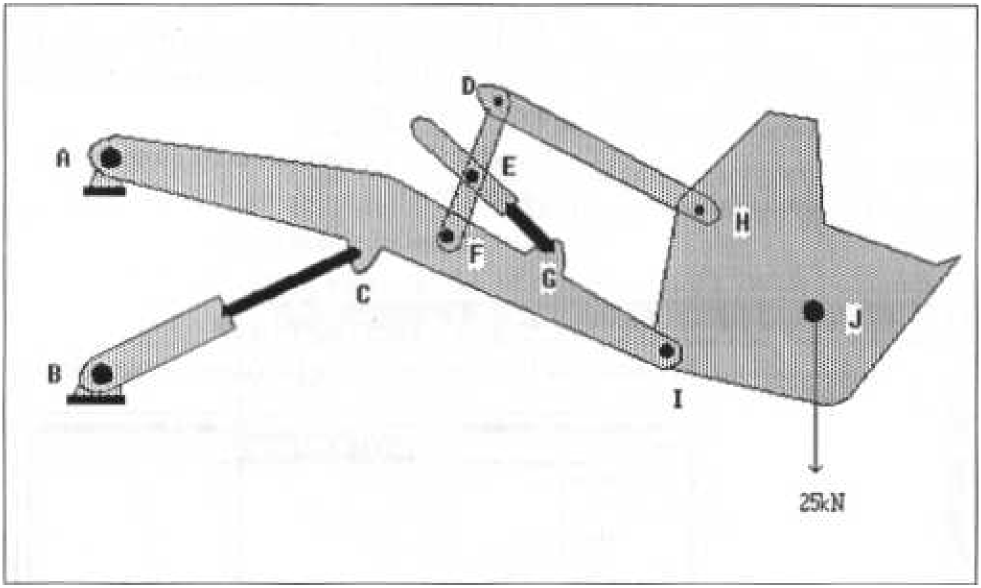
We see that the reaction force increases dramatically



as BE gets larger.

When BE reaches about 85.3, the mechanism becomes ill defined geometrically, and the reaction force becomes infinite.

Example 5: Statics of Hydraulic Actuators



The Problem

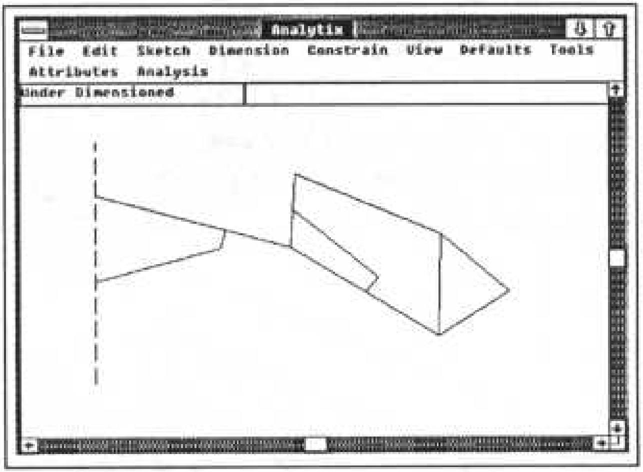
We wish to determine the forces in the hydraulic actuators BC and GE when AF is at 20 degrees to the horizontal and HI is vertical. We also wish to measure the forces which act on member DF.

A is fixed 1.25m vertically above B. Length AF is 2m. Point C is 0.15m below the line joining A and F, and 1.6m along that line from A.

DF measures 0.8m and EF measures 0.5m. FI is 2m and point G is 0.15m above the line joining F and I and 1.2m along the line from F. Angle AM is 170 degrees.

DH is 2m long, HI, IJ and HJ are all lm.

The Model



We draw a vertical construction line on which to locate the point A and B.

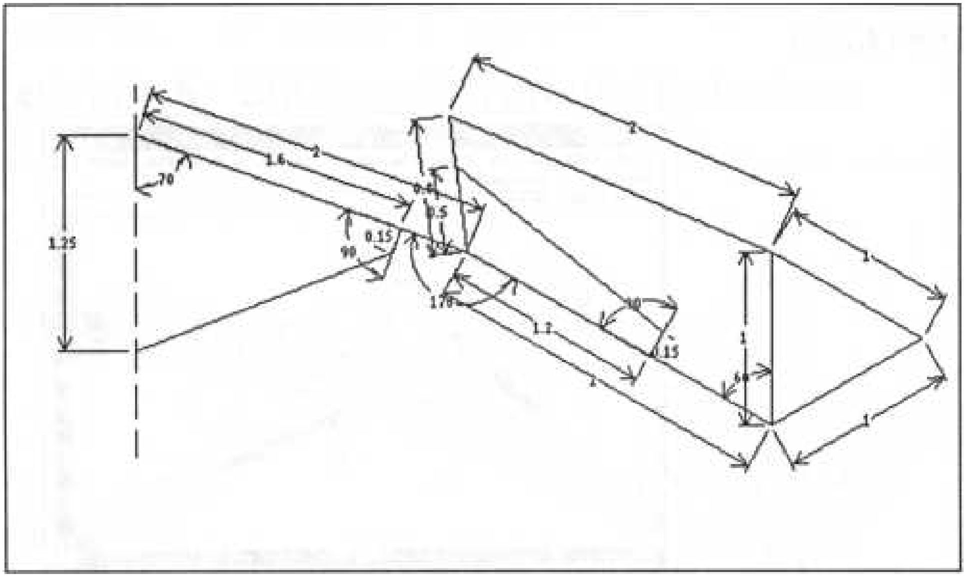
We draw member AI as the pair of lines AF and FI along with two small spurs connecting points C and G to these lines.

The scoop is modeled by the triangle HIJ.

We add the length dimensions given above and 90 degree angles fixing the directions of the small lines at points C and G.

To complete our static model, we need to add the lengths of the actuators BC and GE. However these are not given to us, instead we are given the angles BAF and (indirectly) FIH.

We could sit down and do some trigonometry to, calculate the resulting lengths of BC and EG, but it is far easier to let Analytix do this for us. We can enter the angles into the sketch, let Analytix turn the sketch into a scale drawing then measure the lengths of BC and EG.



AF is 20 degrees from horizontal, hence angle BAF is 70 degrees. IF is 170 degrees from AF, and hence is 30 degrees from horizontal and HI is to be set vertical; so angle HIF is 60 degrees.

We enter a 70 degree angle at A and a 60 degree angle at I.

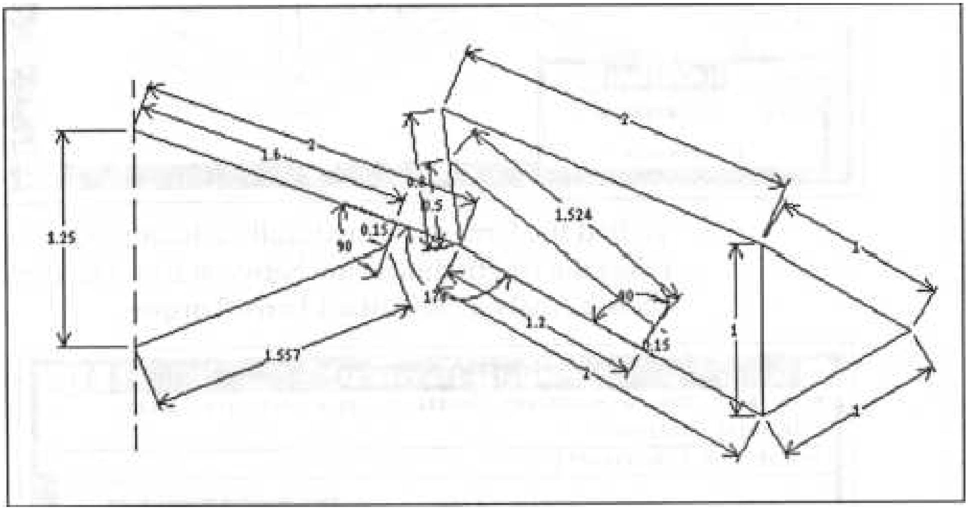
We now have a consistently dimensioned figure, but it is not an accurate model of the statics of the mechanism. This is because the two angles which we just entered have no corresponding physical structure acting to constrain them. Both A and I are pinned joints. The angles there are constrained not by the local physical structure (as they would, for example if they were welded) but indirectly as a result of the hydraulic actuators which constrain lengths BC and EG.

To make an accurate model of the statics, then we must delete the angles at A and I and add length dimensions for BC and EG.

First we measure the lengths of these members which yields the required angles. We do this by selecting the line. The information box at the top of the Analytix window gives you the length of the line.

BC has length 1.557 and EG has length 1.524.

The picture obtained by removing the two angles and adding the two lengths should be identical.

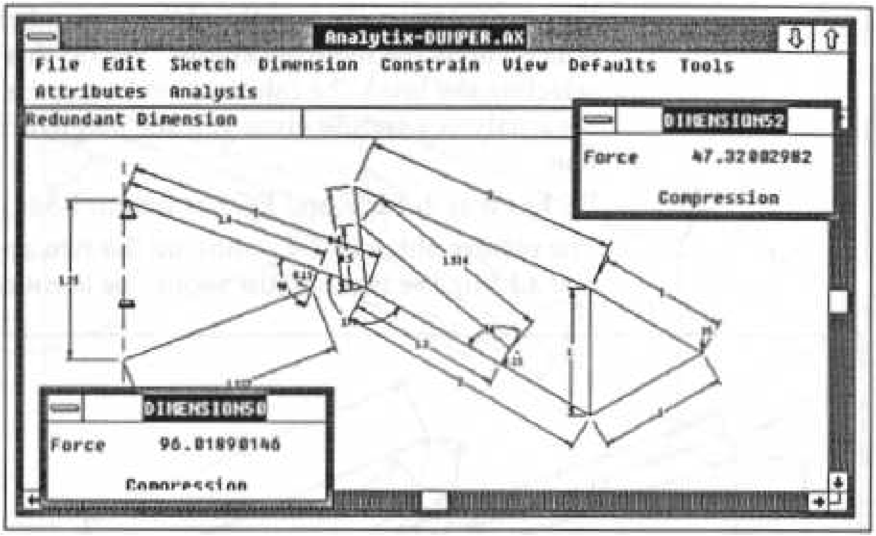


However from the point of view of Statics Analysis they are completely different. BC and EG will now bearloads.

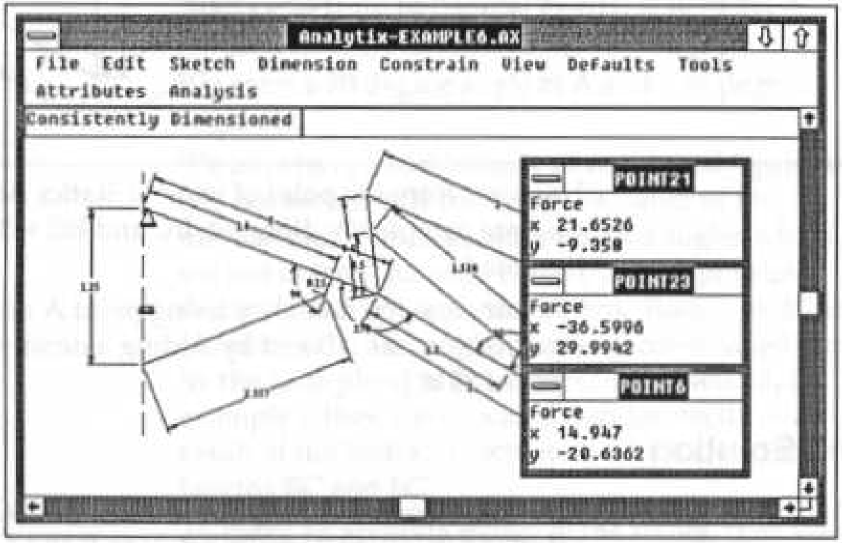
We complete the model by fixing point A and the construction line AB and by adding a downward force of 25 at J.

The Solution

To find the force in the hydraulic actuator BC we select the length dimension between B and C, then choose Analysis/Resultant Force/Torque.

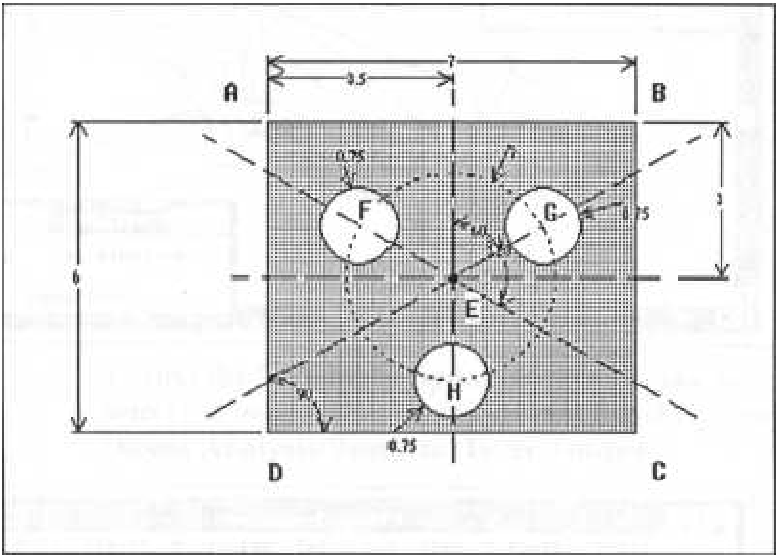


To find the force in the hydraulic actuator EG we select the length dimension between E and G, then choose Analysis/Resultant Force/Torque.



To find the forces on member DF, we select the line DF, then choose Analysis/Force on Pin. We then point in turn to points D, E and F. The force exerted on the member at these points by the other parts of the structure are displayed.

Example 6: Tolerance in Manufacturing



The Problem

During the manufacturing process, the above plate is to fit on a set of three pins whose centers are aligned with the true position of holes F,G, and H. The pins have radius 0.7.

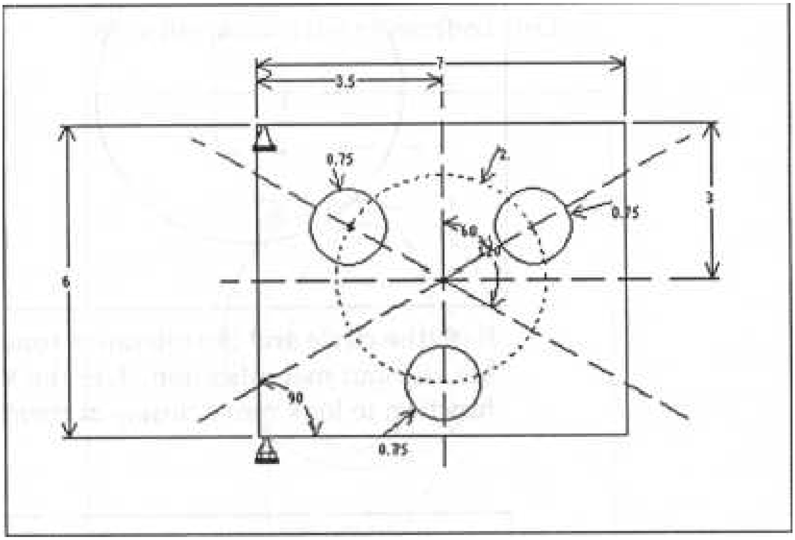
If plate ABCD is aligned by fixing point A and line AD, then we wish to examine whether the holes are guaranteed to fit under the following two sets of conditions:

1 - All linear dimensions have tolerance plus or minus 0.01, all angular tolerances are plus or minus 0.1 degrees.

2 - All linear dimensions have tolerance plus or minus 0.025, all angular tolerances are plus or minus 0.5 degrees.

If the the fit is not guaranteed under absolute tolerancing, we wish to know whether root sum squared statistical tolerancing gives us a fit.

The Model



We draw the part and dimension it as given.

A is made the fixed point and line AD the fixed line.

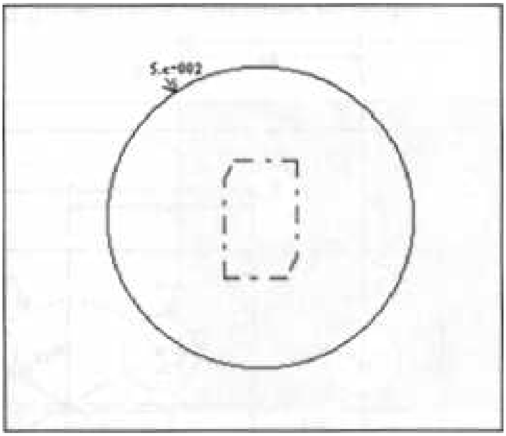
Using the Defaults/Default Tolerance menu option, we set the linear tolerance default to be 0.01 and the angular tolerance default to be 0.1.

The Solution

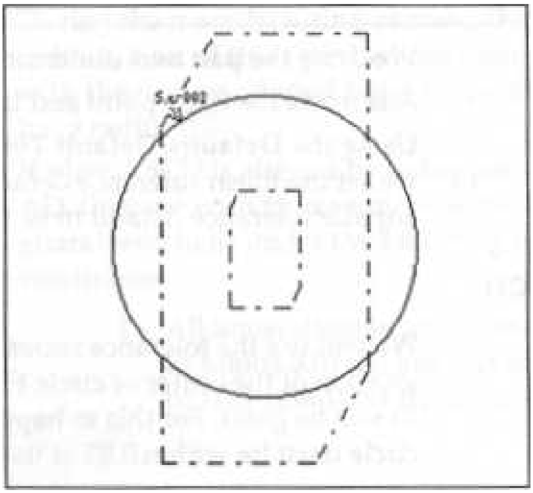
We will use the tolerance zones derived for the location of the center of circle F to tell us whether the fit will be good. For this to happen, the center of the circle must lie within 0.05 of its true position.

We draw a tiny circle of radius 0.05 centered at the center of F. The condition for the fit to be good is that the tolerance zone should lie within this circle.

To create the tolerance zone, we select the point then pick Analysis/Tolerance Zone from the menu.



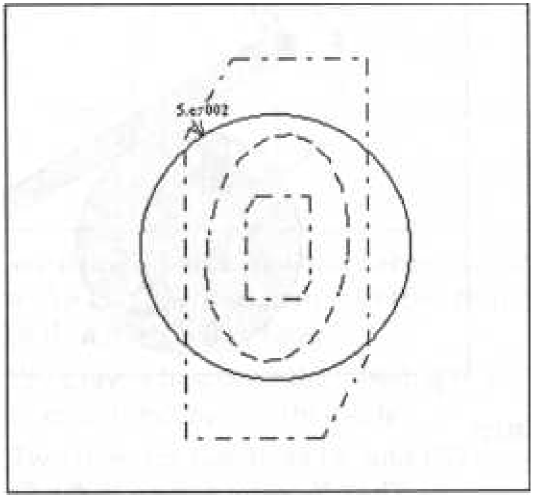
Both the circle and the tolerance zone are too tiny to see without magnification. Use the View/Zoom Box function to look more closely at them.



We see that the tolerance zone does indeed lie inside the circle, so with this tolerance setting, we are OK.

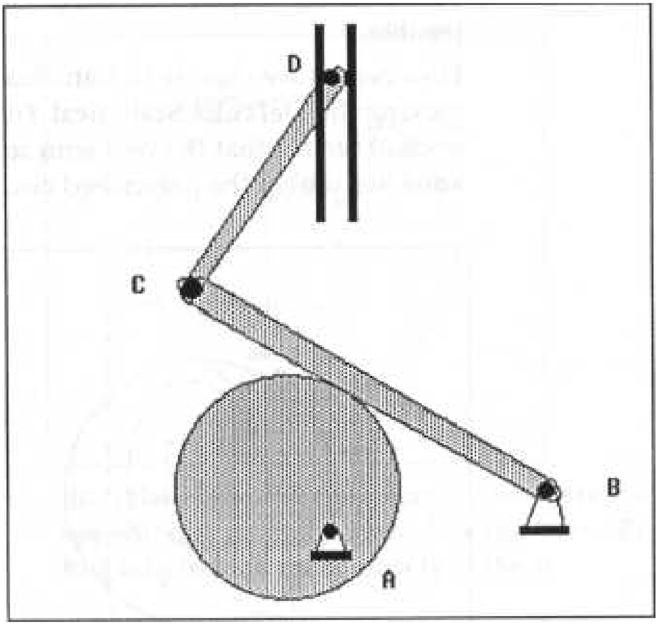
However, if we change the default tolerances to 0.025 for linear and 0.5 for angular, the tolerance zone no longer lies within the circle and we are in trouble.

However, if we change to statistical tolerancing (by picking the Defaults/Statistical Tolerancing menu option) we see that the root sum squared tolerance zone lies within the prescribed circle.



This means if we assume that the individual tolerances are met (say) 99.5% of the time, then the hole will fit at least 99.5% of the time.

Example 7: Kinematics of a Cam



The Problem

The off-centered cam in the figure rotates about A at 50 rad/s. Bar CB maintains tangential contact with the cam. Point A lies on the vertical line on which D is constrained to slide.

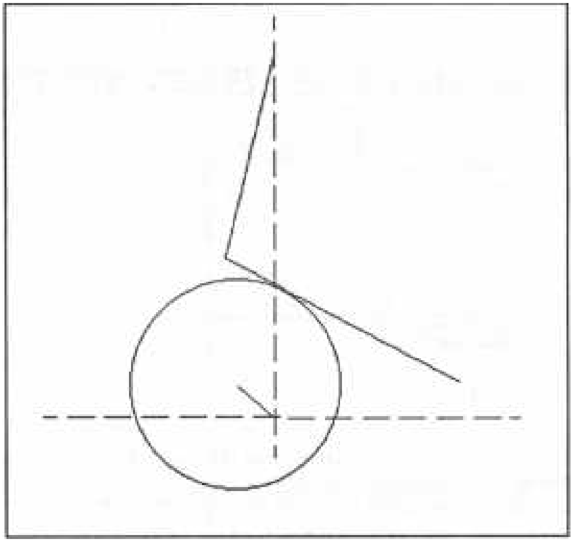
We seek the displacement, velocity and acceleration of D over a complete revolution of the cam.

The cam is radius 2. Point A is distance 1 from the center of the cam.

B is 3.5 to the right of A and 0.75 above it.

BC is length 5 and CD is length 4.

The Model



We draw a horizontal and vertical construction line and a circle whose center is offset from the crossing of the construction lines.

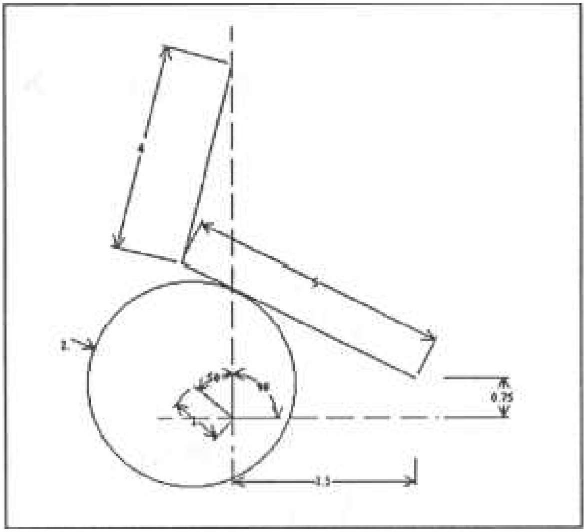
We draw a line from the crossing of the construction lines to the center of the circle.

Two lines for members BC and CD complete the sketch.

We dimension the lengths of BC, CD and the line joining the center of the circle to A. We dimension the distance of B from each construction line and the angle between the construction lines.

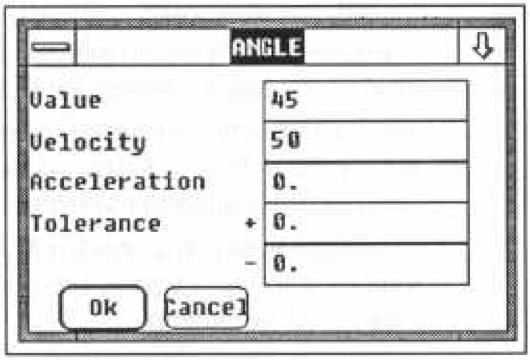
Setting the radius of the circle and making BC tangential to the circle leaves us only to add a driving dimension to move the mechanism.

The angle between a construction line and the line joining A to the center of the circle can be used to drive the mechanism..



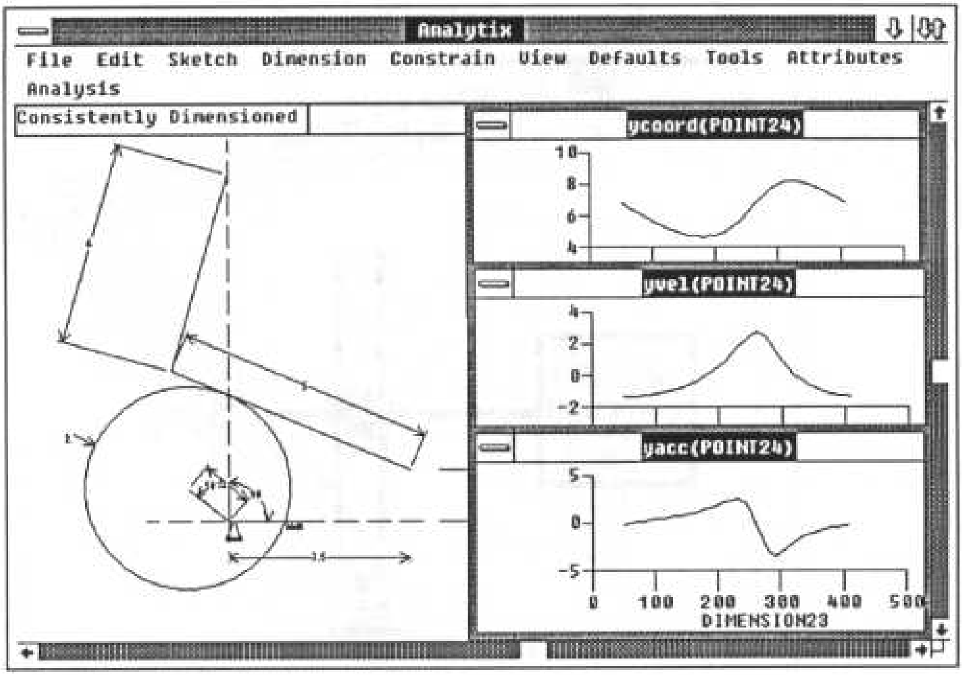
To complete the model, we set point A and the horizontal construction line to be fixed.

It is a good idea at this stage to do an animation of the mechanism to make sure everything looks right. If you have dimensioned the figure inappropriately or forgotten to fix a point and line, this will show up immediately in an animation.



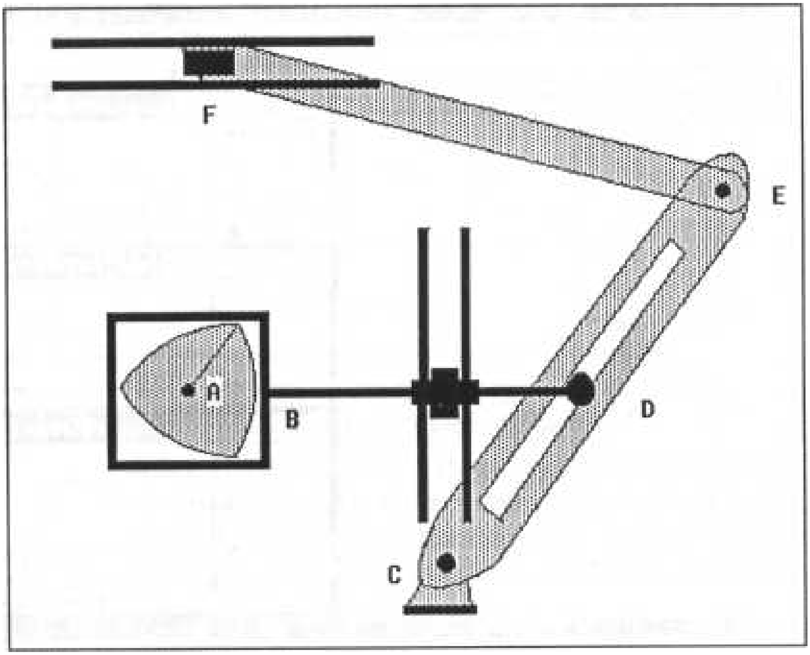
The Solution

Now we need to set the velocity of the driving angle. Use Attributes/Info:



Draw graphs of displacement, velocity and acceleration of point D, using functions ycoord(), yvel(), and yacc().

Example 8: Kinematics of a Cam 2



The Problem

The constant width cam centered at A rotates at a constant angular velocity of 35 rad/s. The cam follower is attached to arm BD which is constrained to remain horizontal.

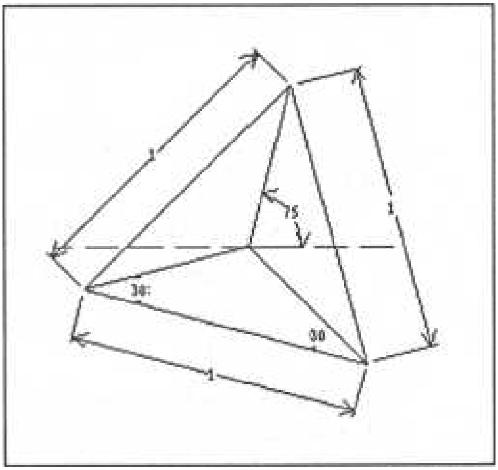
We wish to find the velocity and acceleration of F when one vertex of the cam is at 75 degrees to horizontal (as shown).

The cam has width 1. Point C is distance 1.5 below A and 1.5 to the right. The track which holds F is distance 2 above A. BD and CE are length 3 and EF is length 4.

The Model

First we'll build the cam and cam follower, then we'll add the rest of the linkage.

A constant width cam can be thought of as an equilateral triangle with each side replaced by an arc



with the same radius as the triangle's side length.

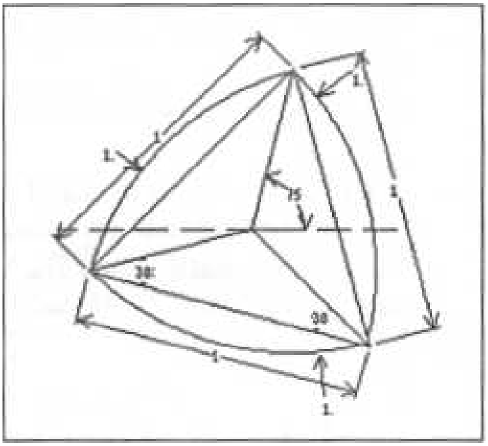
We draw a horizontal construction line to use as a reference for measuring the angle of the cam.

We draw the triangle and its center, positioning the center on the horizontal construction line.

We dimension this figure by specifying the lengths of the triangle sides, the angles of the lines joining the vertices to the center, and the angle one of these lines makes with the horizontal.

We now complete the picture of the cam by drawing arcs between the vertices of the triangle, and giving these arcs a radius of 1.

Our drawing is destined to become rather cluttered. To reduce the clutter, we use the level management



capabilities. First let us put all the dimensions so far on a new level.

I - Pick Edit/Select All Dimensions.

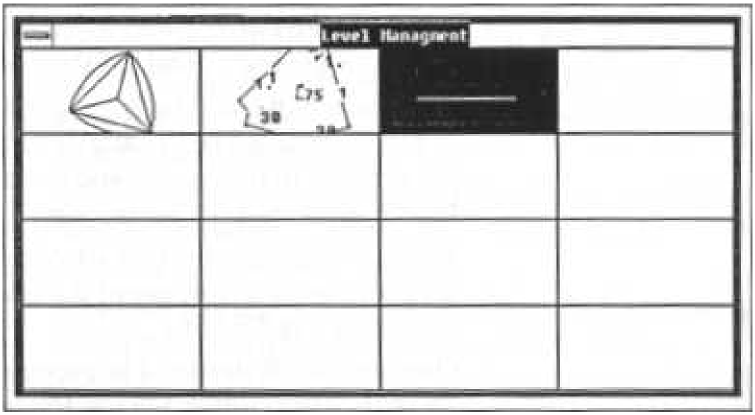
2- Choose View/Change Level.

3 - Click on the box representing the second level.

The dimensions are now all on level 2. We also might want to put the horizontal construction line on yet another level. To do this:

1 - select the construction line.

2- Choose View/Change Level.

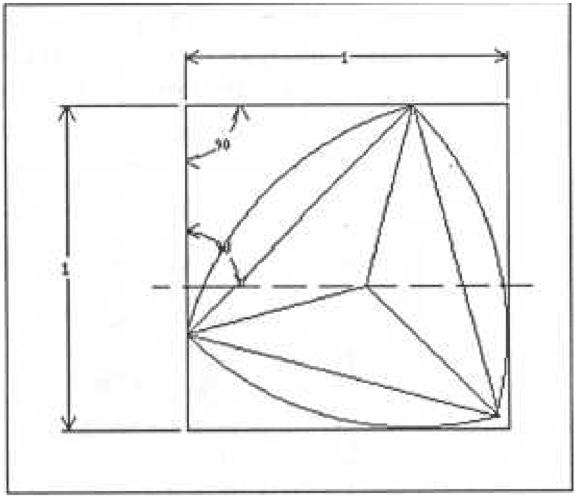


3 - Click on the box representing the third level.

We can now draw the cam follower: a box fitting round the cam.

The follower is dimensioned by setting the parallel distance between its sides and a right angle between two of its sides.

The follower is constrained to sit with its sides aligned with the horizontal and vertical. We can



model this by setting the angle between the side of the follower and the horizontal construction line to be 90 degrees.

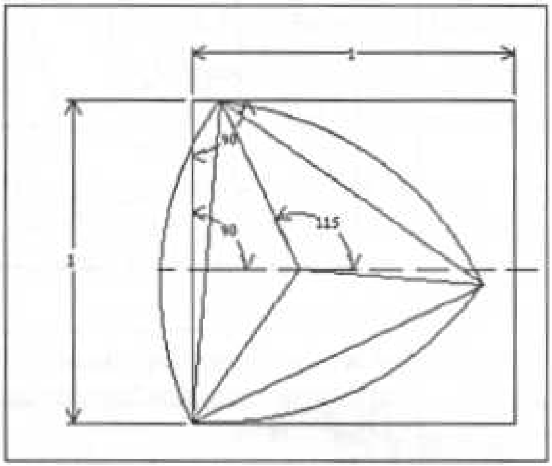
We have now described the shape of the follower and its alignment, we must now describe its contact with the cam.

The cam is always in contact at two vertices and two arcs. There are two issues here.

If you specify both the point contacts and the arc contacts, the system is overdetermined. For example if you set the two vertices to lie on the edges of the follower, then the opposite arcs will automatically be tangent to the opposite sides of the follower. Therefore we only need specify that the two vertices of the cam lie on the edges of the follower.

A more serious problem is that as the cam turns, a different pair of vertices become the contact points with the follower. To model this in Analytix is difficult: you have to go in and break the point-on-line constraints which you just added then add new point-on-line constraints to the new vertices.

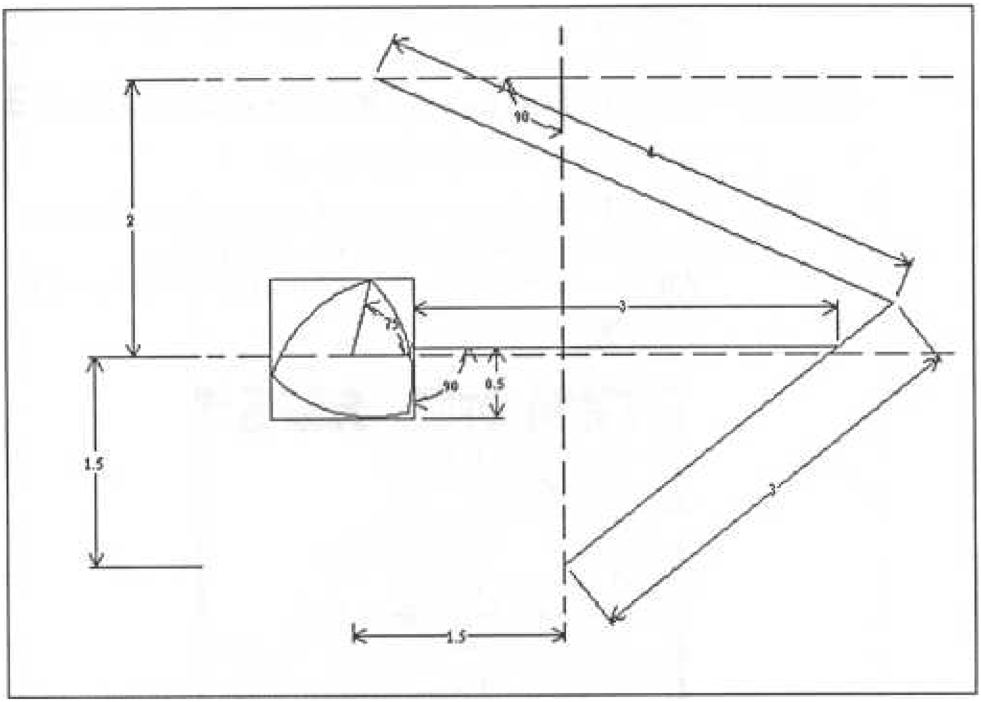
To see the problem try incrementing the 75 degree angle between the cam vertex and the horizontal. You will see that the model is fine until this angle reaches 90 degrees, then the arc between the two



contact points starts to leave the follower box. At this stage, we need to set the other point in the figure to be in contact with the follower.

Our model, therefore, is is only good for a segment of the total revolution of the cam. Specifically, for the segment between 60 and 90 degrees.

We now enter the rest of the mechanism. We draw a horizontal construction line for F to run on, and a



vertical construction line to position C on. We draw the linkages CE, EF and BD.

A parallel distance dimension is given between the two horizontal construction lines, and a right angle between the vertical and horizontal construction lines. The distance between C and the horizontal line through A is given and the distance between A and the vertical line through C.

Lengths are given for the three links and link BD is specified relative to the follower by setting its angle with the follower's edge and the distance of B from the corner of the follower.

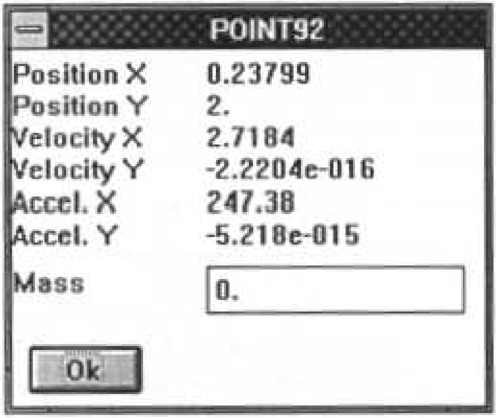
The Solution

Our geometrical model of the mechanism is now complete. To perform the required kinematic analysis, we simply need to give a velocity to the cam angle.

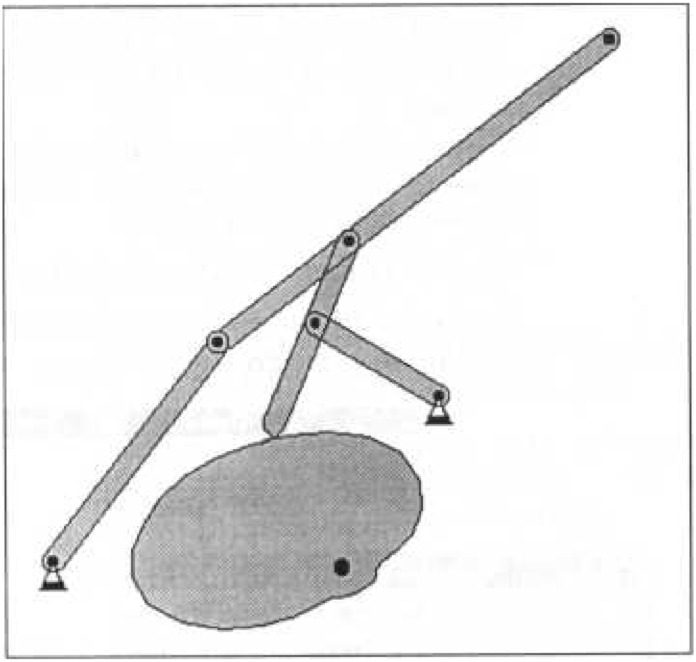
Select the 75 degree angle, then pick Attributes/Info.

Enter 35 for the velocity of the angle.

To find the instantaneous velocity and acceleration of point F, we select the point, then pick Attributes/Info from the menu (shortcut by double clicking on the point)..



Example 9: A Cam Driven Crosby Linkage



The Problem

A cam is frequently described by a displacement function which is entered into Analytix as a formula for the value of a dimension. The type of dimension will depend on the type of follower; it will be a length if the follower is reciprocating, it will be an angle if the follower is oscillating.

One aspect of cam displacement functions is that they are typically made up of different segments, each of which has an analytical formula. In this example we model a cam with two dwells joined by simple harmonic rises.

The Model

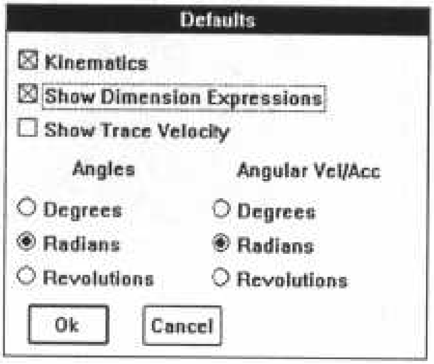
The cam of length d has displacement 1.5+sin(2*t-90) if t is between 0 and 90 degrees, 2.5 if t is between 90 and 180 degrees, 1.5+sin(2*t-270) if t is between 180 and 270 degrees, and 0.5 if t is between 270 and 360 degrees.

We want to use derivatives of cam displacements in order to study velocity and accelerations, so it is convenient to work in radians:

Select the Defaults / System Defaults menu option.

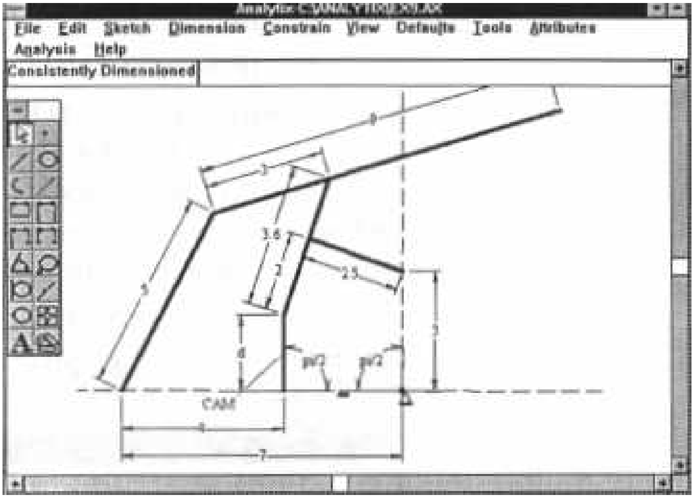
Notice the default angular units are degrees.

Click the Radians button under Angles. Then select OK.



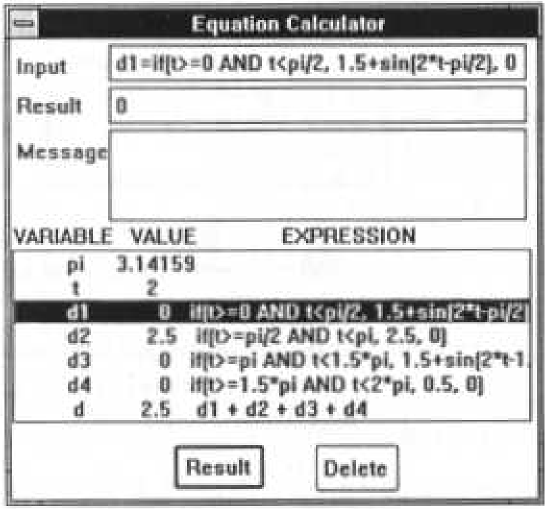
Now sketch the linkage as shown below giving the line length representing the distance between cam and follower some initial value, *e.g.* d=1.

We will use the Calculator to define distance, velocity, and acceleration in the cam's four quadrants.



Select the Tools / Calculator menu option.

Enter the value for pi, an initial value for t, and four displacement formulas in the Input box. Click on the Result button after entering each formula.



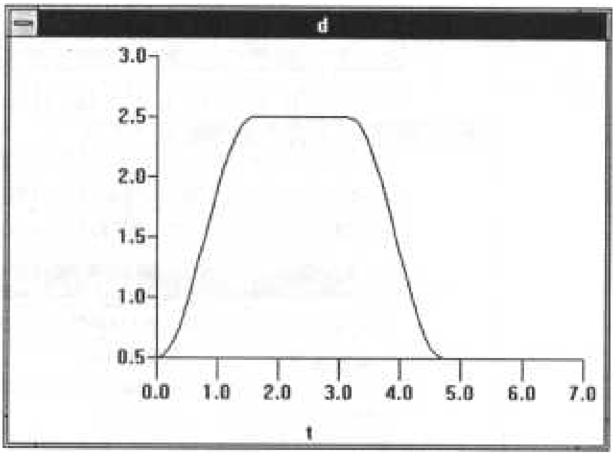
Notice the use of the if function to define the quadrants:

```
dl = if(t>=0 AND t<pi/2, 1.5+sin(2*t-pi/2), 0)
d2 = if(t>=pi/2 AND t<pi, 2.5, 0)
d3 = if(t>=pi AND t<1.5*pi,
          1.5+sin(2*t-1.5*pi), 0)
d4 = if (t>=1.5*pi AND t<2*pi, 0.5, 0)
```

The final formula you see in the expression list calculates the current value of d.

```
d=dl+d2+d3+d4
```

We can now display a graph of d against t, which is a displacement diagram for our cam.



In order to generate a kinematic model of our cam, we need expressions for the velocity and acceleration of the cam displacement.

Let dt be the (constant) angular velocity of the cam in radians per second. If the cam is rotating at 3 revs per second, then:

```
dt = 6*pi
```

Now we create expressions v1, v2, v3, v4 and al, a2, a3, a4 for the velocity and accelerations of the different segments of the cam.

v1 = 2 * cos(2*t - 0.5*pi) * dt

v2=0

v3 = 2 * cos(2*t -1.5*pi) * dt

v4=0

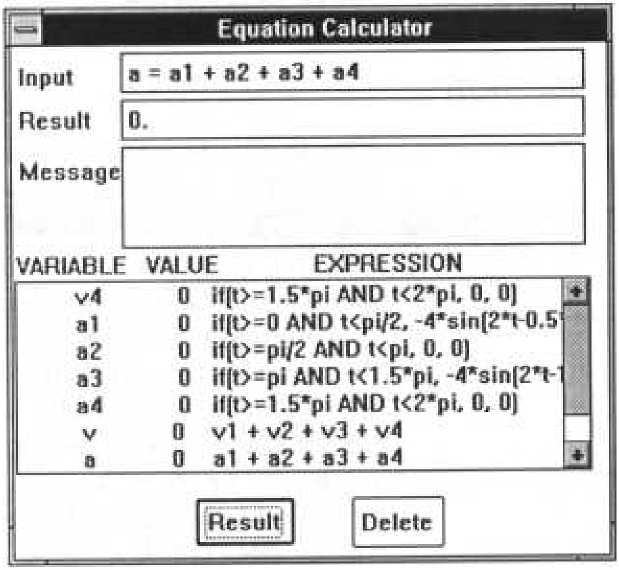
al = 4 * sin(2*t - 0.5*pi) * dt

a2=0

a3 = 4 * sin(2*t -1.5*pi) * dt

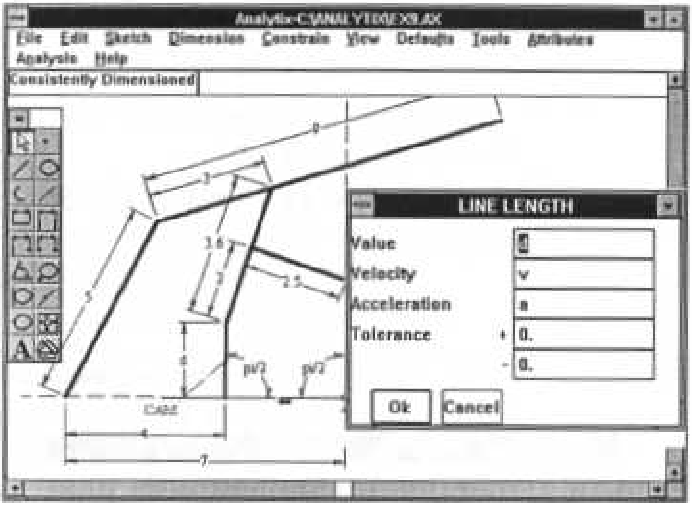
a4=0

The remaining two expressions to enter are the summing of the Vs and a's as shown in the calculator window below.



Having input these calculations, we must now assign v and a to the dimension for the cam:

From select mode, double click on dimension d and you will see the Line Length info box. Enter v and a into their respective boxes.

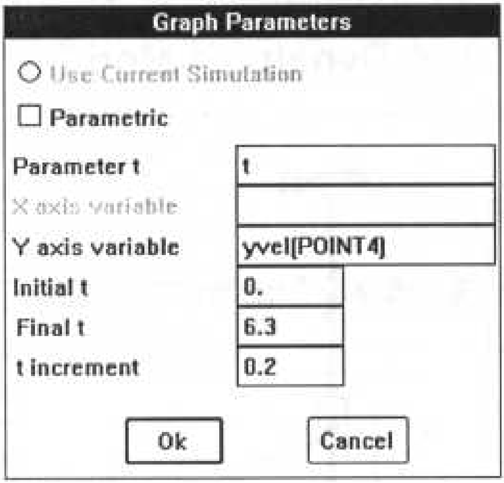


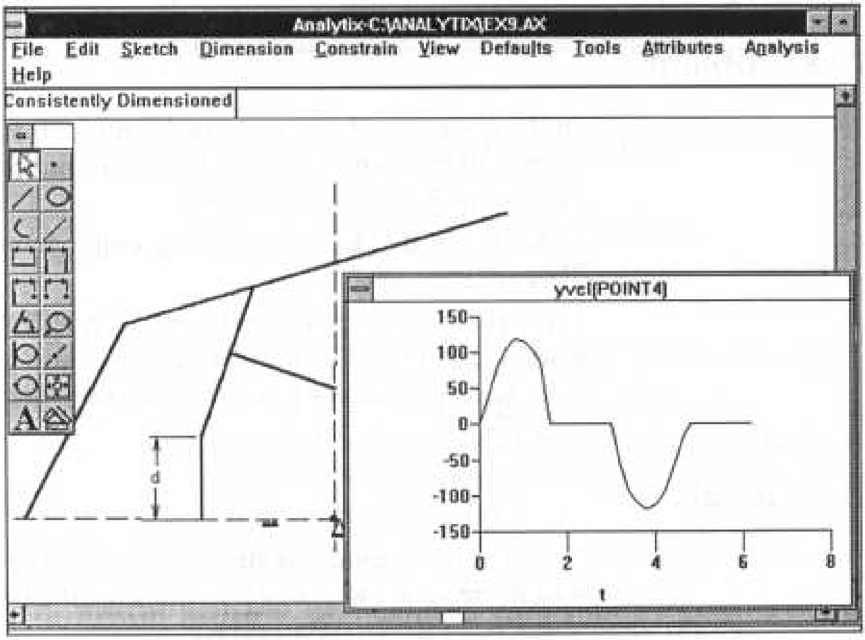
The Solution

We can now examine the kinematics of the linkage. The cam drives a Crosby linkage, which produces an amplified approximate straight line motion.

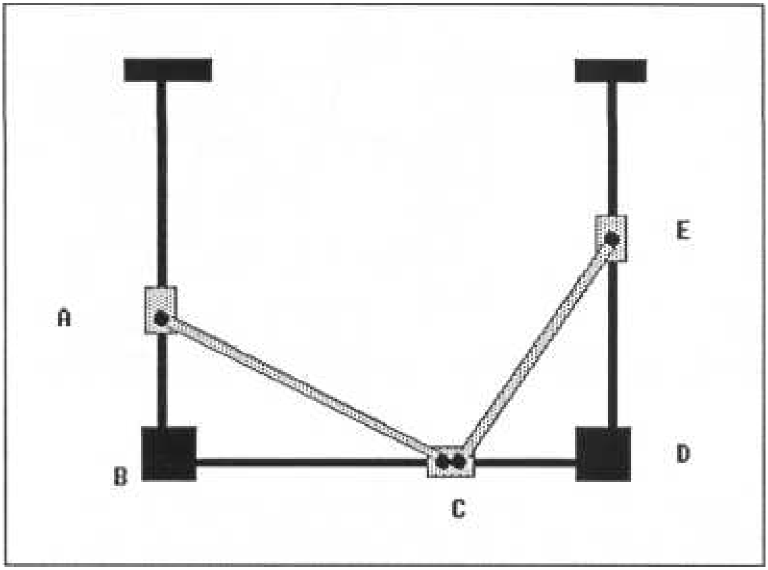
We have given the length dimension for the cam a value of d, a velocity of v and an acceleration of a. The output graph depicts the vertical component of the resultant velocity of the end effector. To do this we use the function yvel(pointxx) for the Y axis variable, and as with the displacement plot above, parameter t goes from 0 to 2"pi. As shown below, the initial t, final t, and increment must benumeric values, not expressions.

Note that the geometry of the cam is not explicitly created. Instead, the kinematic behavior of the cam / follower pair is embodied in the formulas for d, v, and a.


Graph Parameters
Use Current Simulation
Parametric
Parameter t
t
X axis variable
Y axis variable
yvel[POINT4]
Initial t
0.
Final t
6.3
t increment
0.2
Ok
Cancel


Analytix-C:\ANALYTIX\EX9.AX
File Edit Sketch Dimension Constrain View Defaults Tools Attributes Analysis
Help
Consistently Dimensioned
d
yvel[POINT4]
150
100
50
0
-50
-100
-150
0
2
4
6
8
t

Example 10: A Dynamics Model



The Problem

In the above mechanism, bars AC and CE are each length 400mm and have negligible mass. The vertical rails are 560mm apart.

Collars A and C have mass 200g, collar E has mass 1008.

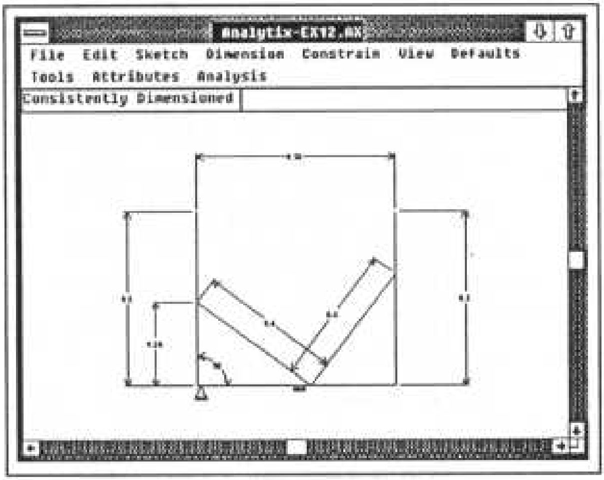
If the mechanism is aligned vertically, and A is released from rest 240mm above B, we are to find the initial acceleration of A and the forces in bars AC and CE.

The Model

The geometric model is simple: lines for the vertical and horizontal rails, and two lines for the bars AC and CE.

We dimension the geometry by giving the parallel distance between the two vertical rails, entering the right angle between vertical and horizontal, giving lengths to the vertical rails, and specifying the lengths of AC and CE.

Our dimensioning is completed by specifying the distance between point A and point B.



To set up the model for performing the dynamic analysis we need to:

1 - Set the fixed point and line.

2 - Make sure the units being used are correct.

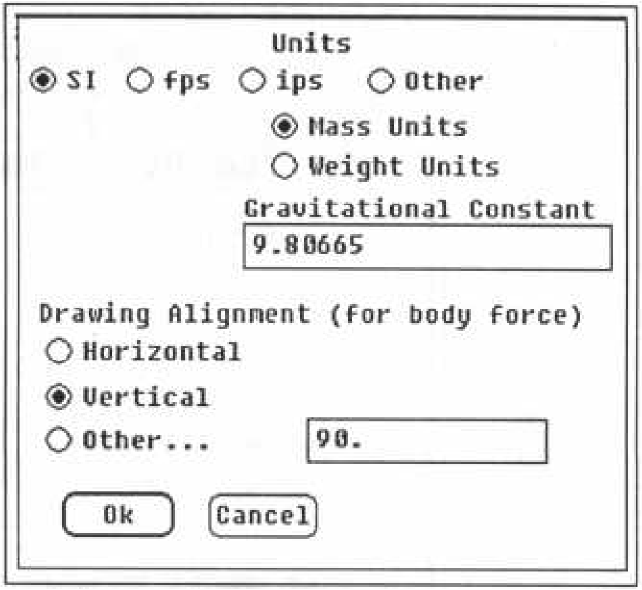
3 - Specify the orientation of the drawing.

4 - Specify masses for A, C and E.

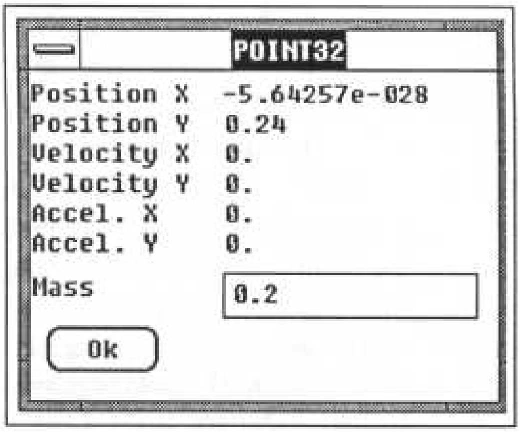
Fix point B and the horizontal rail by selecting both the point and the line and using the Constraints/Fix Point/Line menu option.

To ensure our units are correct and specify the orientation of the drawing, we use the Defaults/Dynamics Defaults menu option.

Set the units to be SI and the drawing alignment to be Vertical by clicking on the appropriate buttons of the Dynamics Defaults dialog box.



Set the masses of the collars at A, C and E by selecting each point in turn and picking the Attributes/Info menu option.



Enter the mass in the appropriate box and click on the Ok button.

The Solution

We now have a model which knows about masses and therefore the gravitational forces applied to those masses. We wish to derive the acceleration of A due to those forces.

Our model, however is not an accurate representation of physical reality, because we have a dimension keeping point A 240mm above point B. In reality there is nothing constraining A to stay in this location.

Further the dimension between A and B is supporting a load. In fact we have a model of the situation before A is released from rest.

To model the situation an instant after A is released, we need to give the dimension between A and B an acceleration.

But how do we know what acceleration is correct?

The answer is we don't but we will find out. Here's how:

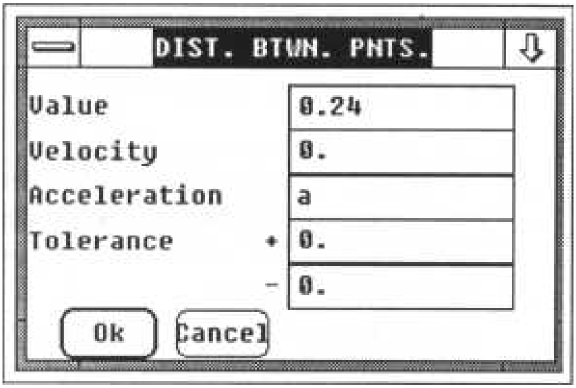
t - We set the distance between A and B to have acceleration a (a variable).

2 - We use the Iteration tool to find the value for a which leaves the dimension supporting no load.

This is now an accurate model of the physical reality an instant after A is released.

To set the acceleration of the distance between A and B select the dimension and pick Attributes/Info from the menu.

You now see the Dimension Info Box. Enter a as the acceleration.



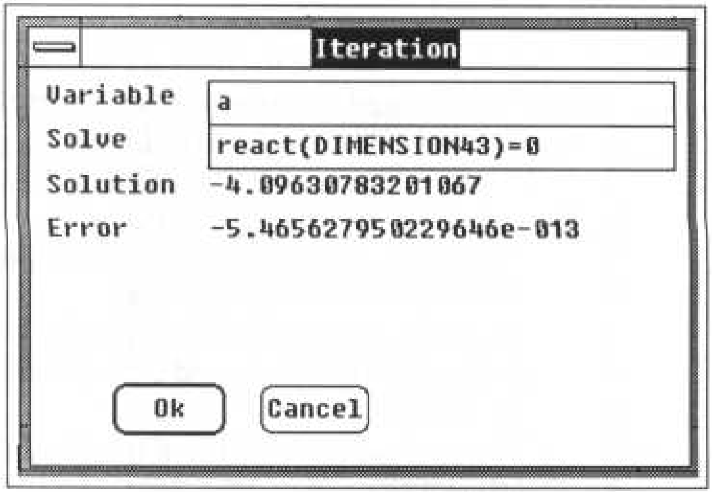
To perform the iterative solution choose Tools / Univariate Iteration from the menu.

You will see the Iteration Box. Type a as the variable to be iterated on.

Type react(in the Solve box. Now go into the main drawing and select the dimension between points A and B. The name of the dimension will be inserted into your equation (in our case DIMENSION43).

Finish off the equation react(DIMENSION43)=0.

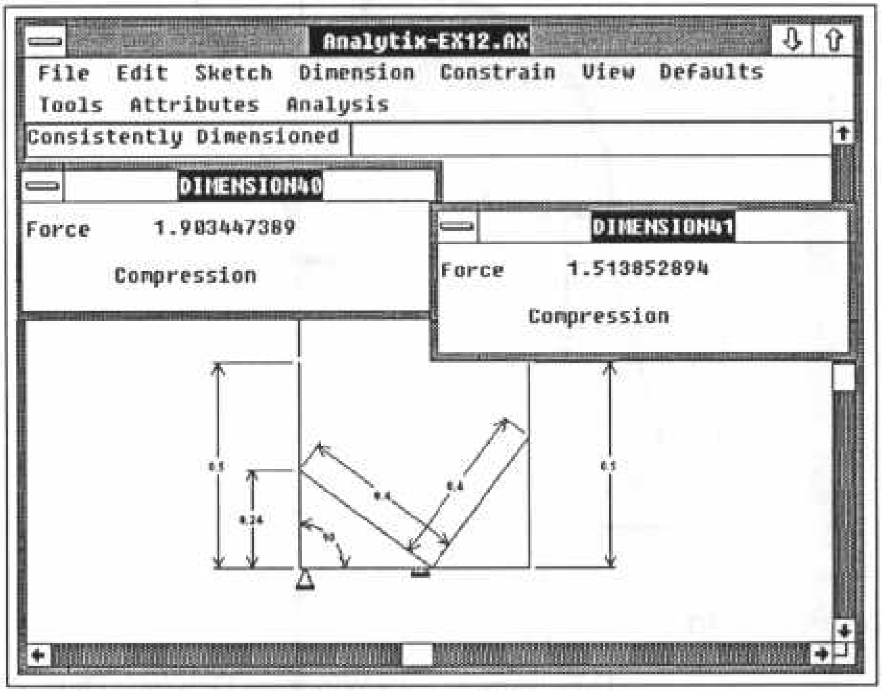
Now Click on the Ok button to solve the equation.



The solution yields that this length dimension has an acceleration of -4.096 ms^-2. As point B is fixed, this is the acceleration of point A.

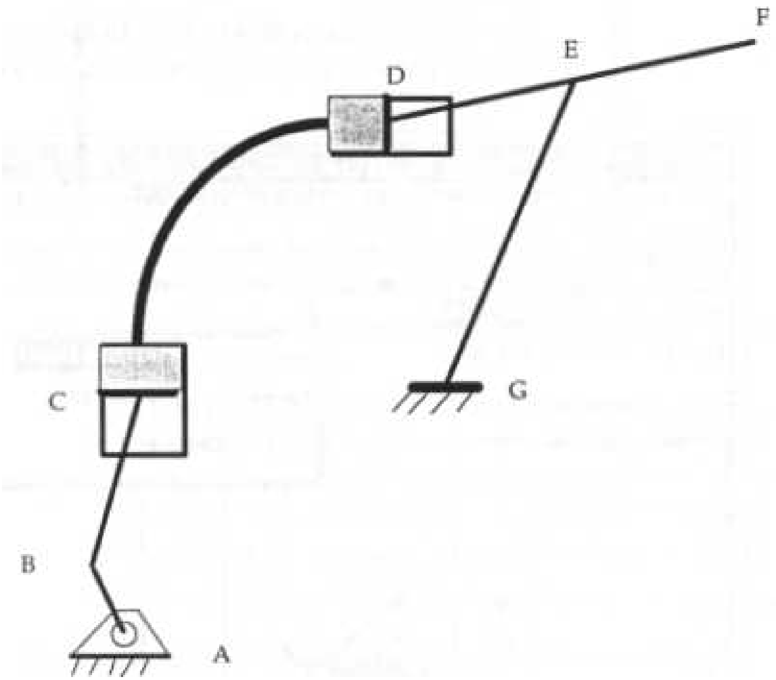
We now need to find the reaction forces in bars AC and CE.

We do this by selecting in turn the length dimension of each bar and choosing the Analysis/Resultant



Force/Torque.

Example 11: Dynamic Data Exchange

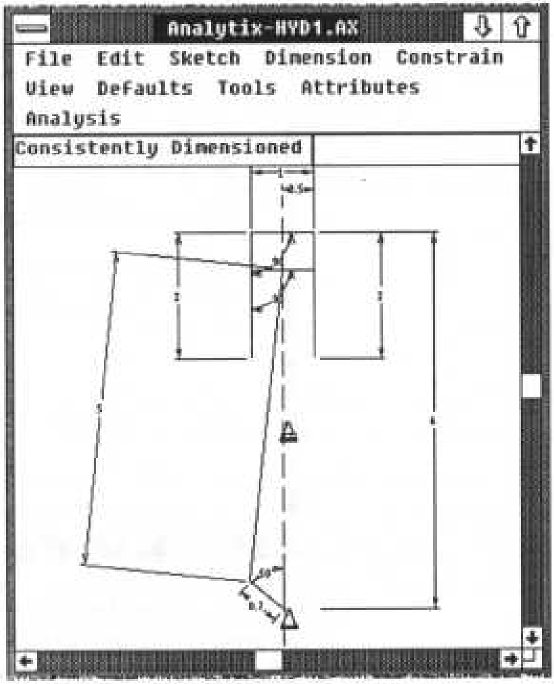


The problem

This example uses Windows Dynamic Data Exchange (DDE) to model the above hydraulically linked mechanism. Crank AB drives piston BC, which forces fluid out of the cylinder C and into the cylinder D. The piston in cylinder D is driven by the consequent fluid flow and in turn drives mechanism DEFG. The cylinders have different widths.

The Model

To analyze this system, we create two separate models, one of the input crank and cylinder, one of the output mechanism. These models are created in separate instances of Analytix. We link the two models by a Dynamic Data Exchange (DDE) connection between the two copies of Analytix which contain the two models.

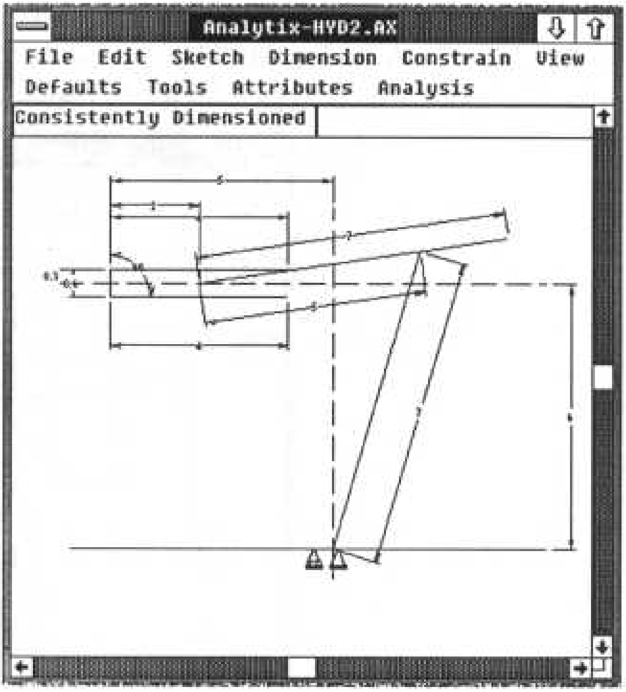


The first model is of the input cylinder. Create this in one copy of Analytix. The variable which will be output is the volume of fluid in the cylinder. Enter this as the formula:

v = 3.14159*0.5A2*distance(point15,line18)

Where point15 is the point at the center of the piston and line18 is the top of the cylinder. 0.5 is the radius of the cylinder. (If you wished, you could parametrize the problem by entering the variable r for this radius.)

To create a second model, do not clear the current version of Analytix, but start up a second copy. Draw the output mechanism as shown.

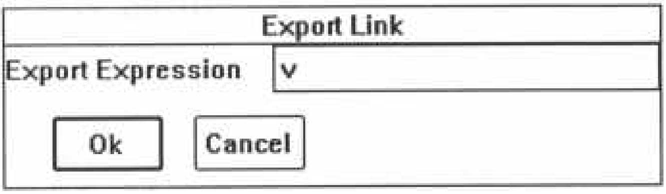


The position of the piston in the cylinder is given by a parallel distance of 1. This is temporary and will be altered when we connect up the two drawings.

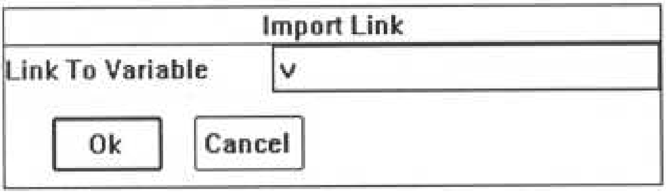
The next step is to create a DDE link between the two models along which we can pass the value of v. Then we can cause the position of the piston at D to have the correct behavior as the volume in piston C changes.

To create the link, go into the input model, and select Edit / Export Link. You will see the Export Link

Dialog Box. Here you should enter the expression whose value you wish to export: in this case v.



Now go into the output mechanism model and select Edit / Import Link. You will see the Import Link Dialog box. You should enter here the name of the variable where you want to store the incoming value. In our case we'll call this v also.



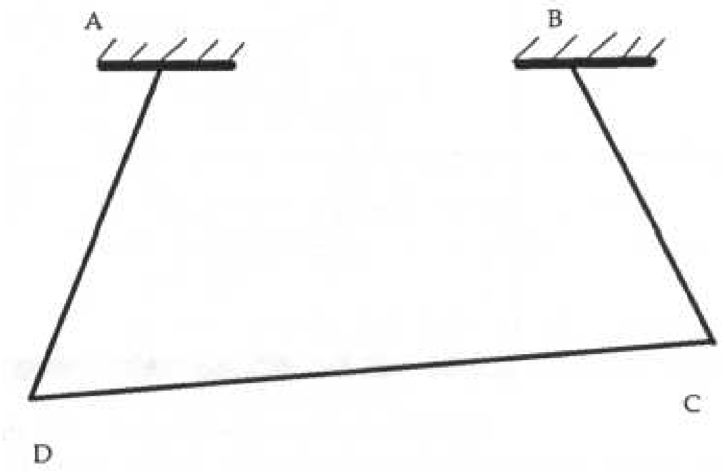
Now the variable v in the second copy of Analytix should have the same value as v in the first copy.

Our next step is to make the piston location D depend on v in the appropriate way. Bring up the Info Box for the parallel distance dimension which specifies the location of D. Enter the formula:
(2-v)/(3.14159*0.3^2)

You should see the piston move to a new location.

Now if you change the value of the crank angle in the input drawing, you should see the output mechanism update accordingly.

Example 12: Steady State



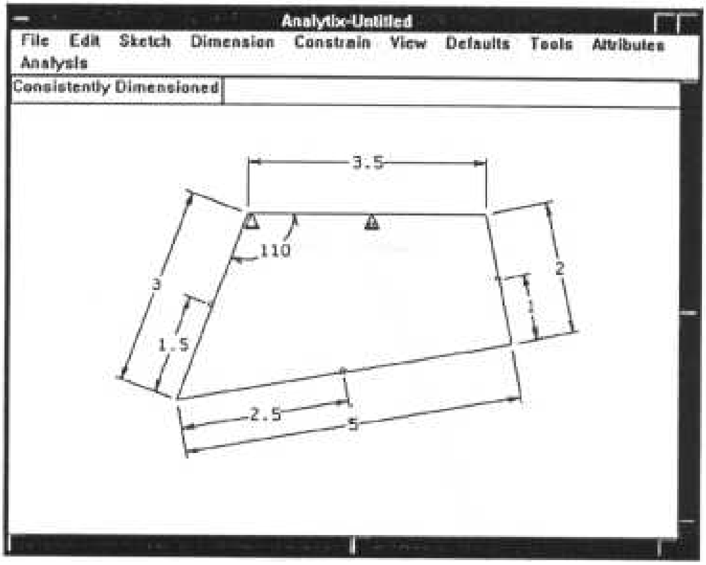
The Problem:

The above linkage hangs vertically under its own weight. Our problem is to find the equilibrium position of the linkage.

AD is 3 meters long, DC is 5 meters, BC is 2 meters and AB is 3.5 meters. All three bars have mass density lkg/metre.

The Model

We draw the linkage in a sample configuration such that angle DAB is 110 degrees. W e add mid-points to each line and give each mid point the mass of its bar. We make sure the drawing is vertical (using the Defaults / Dynamics Defaults menu option.



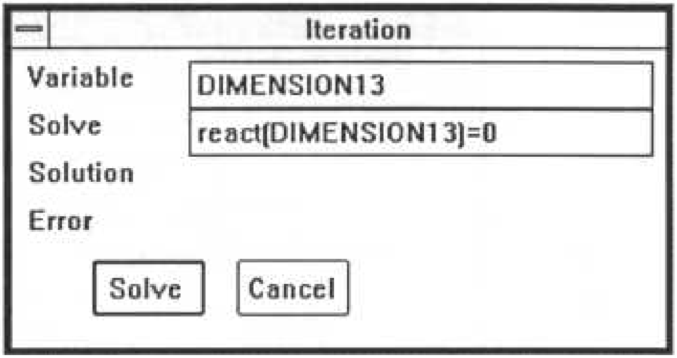
We now have a static model of the situation where the structure is being held in place by the angle in the picture. In fact there is no support at this angle. Hence if the structure were in equilibrium, there would be no force transmitted by this angle.

To find the equilibrium position of the structure, therefore, we need to find a value for the angle such that the force transmitted by the angle is zero. This problem is conveniently solved using the Iteration tool.

The Solution

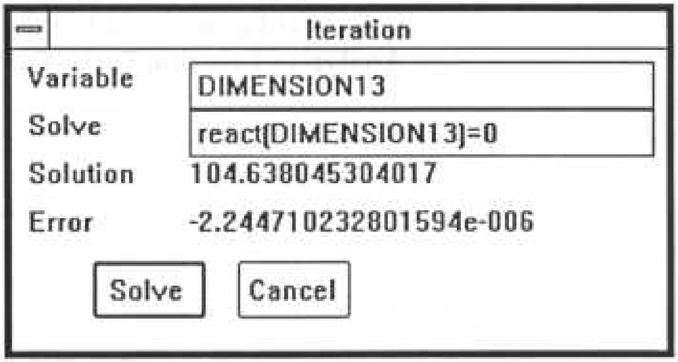
Select Tools / Iteration. You will see the Iteration Box. Click in the Variable Box, then select the angle. The name of the angle should appear in the Variable Box. We now enter the equation to be solved in the Solve Box. This is:

react(DIMENSION13)=0



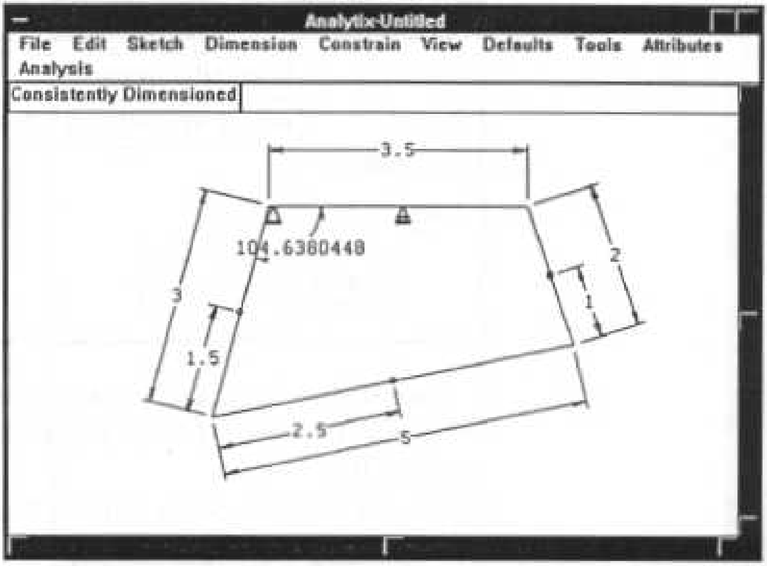
Instead of 13, you should use whatever the dimension number is for the angle dimension in your model.

When you push the Solve button; Analytix will search for a value of DIMENSION13, which satisfies react(DIMENSION13)=0.



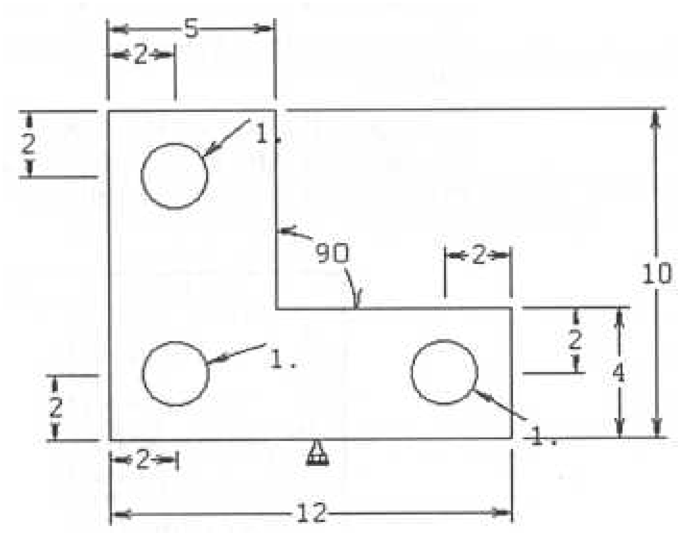
The Solution we see is 104.638045 degrees, and this is accurate to 5 decimal places.

If you redraw the picture by clicking on the Scroll Bar, you will see the model in the equilibrium configuration.



To confirm that no force is transmitted by the angle in this configuration, select the angle, then choose Analysis / Reaction Force/Torque.

Example 13: Area Mass Properties



In this example we see how to display the area and area moments of inertia of the L bracket displayed above.

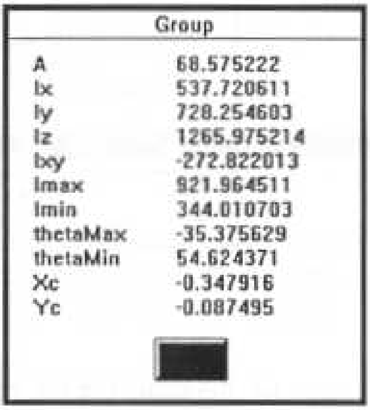
In Analytix, mass properties are attributes of a group of lines and arcs. To display mass properties, therefore, we need first to group the lines, arcs and circles which form the outline of the part under study.

We do this by selecting all the lines which form the outline of the bracket, then selecting the circles which represent holes cut out of the bracket (while holding down the Shift button). Then we select Edit/Group from the menu.

The profile is now a single group. Whenever you select an entity in the group, they will all be selected, and the Current Selection Display in the top right of the Analytix screen will contain the group name and not the name of the individual line or arc.

(To ungroup the entities use the Edit/Ungroup menu option.)

To display the area mass properties, select the group then pick the Attributes/Info menu option. (Alternatively you can double click on the group.)



The following properties are displayed:

Area - the area of the outer profile minus any profiles contained within. (If the profiles intersect the result is meaningless.)

Ix - Area moment about the x-axis through the centroid.

Iy - Area moment about the y-axis through the centroid.

Iz - Area moment about the z-axis through the centroid. (This is sometimes denoted J).

Ixy - Area product of inertia through the centroid.

Imax, Imin - Maximum and minimum area moments about an axes through the centroid.

thetamax, thetamin - the angles which the directions of maximum and minimum area make with the x-axis.

Xc, Yc - the x and y coordinates of the centroid.

Mass Functions

Note that we can also obtain the values of these quantities using the following functions:

area(group1)

Ix(group1)

Iy(group1)

Iz(group1)

Ixy(group1)

Imax(group1)

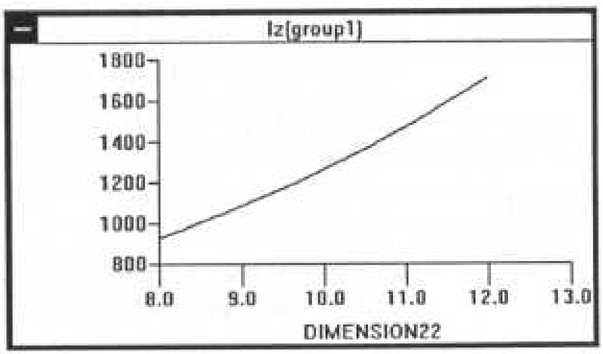
Imin(group1)

Xcentroid(group1)

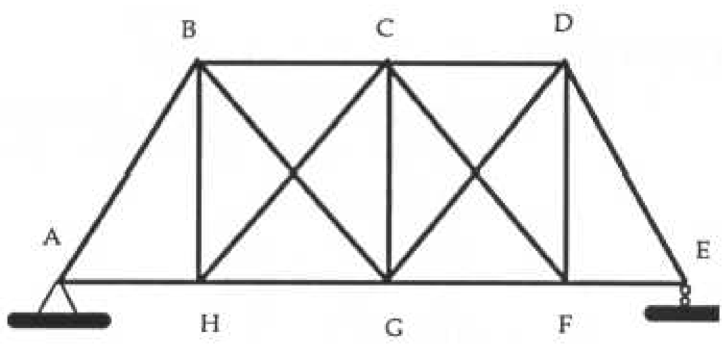
Ycentroid(group1)

Thus we can access the mass properties for use with the Table, Graph, Calculator and Iteration tools.

Below is a graph of Iz varying as the height of the L bracket varies from 8 to 12.



Example 14: Truss Deflection & Stress



The Problem

The above bridge truss has a vertical height of 40ft. Each of the four horizontal spans is 30ft. The horizontal members have a cross sectional area of 15 square inches, the vertical members have a cross-sectional area of 10 square inches. Members AB and DE have a cross-sectional area of 25 square inches and the other diagonal members have cross sectional areas 12.5 square inches. The modulus of elasticity for each bar is 30000 kips per square inch.

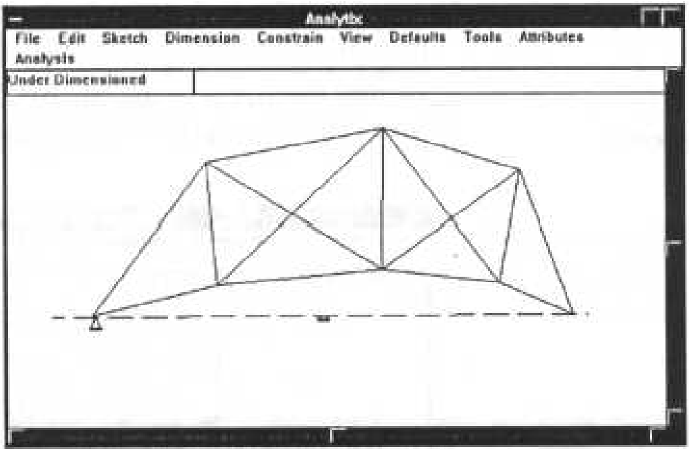
If H, G and F each have a vertical load of 80kips, we wish to find the vertical displacement of G and the stress in AB.

The model

Point A of the bridge is fixed, and point E is free to roll on a horizontal line through A. We use a construction line to represent this horizontal line and draw the bridge truss ensuring that points A and E lie on the construction line but H,G and F do not.

A will be the fixed point, while the horizontal construction line will be the fixed line for the diagram.

We use Defaults/Default Bar Properties to set the



modulus of elasticity of all bars to be 30000. We set the default cross sectional area to be 15.

The cross sectional areas for the vertical and diagonal members may now be set individually using the Info box for each line.

Length dimensions are added to the model to make it consistently dimensioned. (As we have specified the cross sectional area in inches, we specify lengths also in inches.)

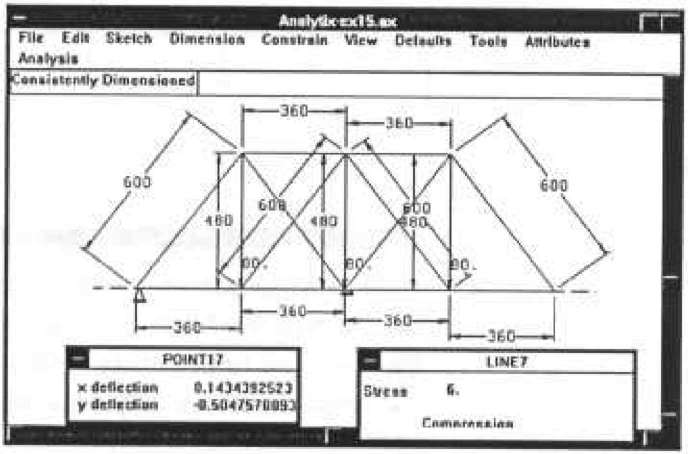
Note: In Analytix the statics model is usually determined by which dimensions are specified. An exception to this is in deflection and stress analysis of trusses. In this case the static model is determined by which lines have non-zero modulus of elasticity and cross sectional area.

As the truss we are analyzing is statically indeterminate, we will not be able to dimension the length of all the bars. However we simply add enough dimensions to fully specify the model.

The solution

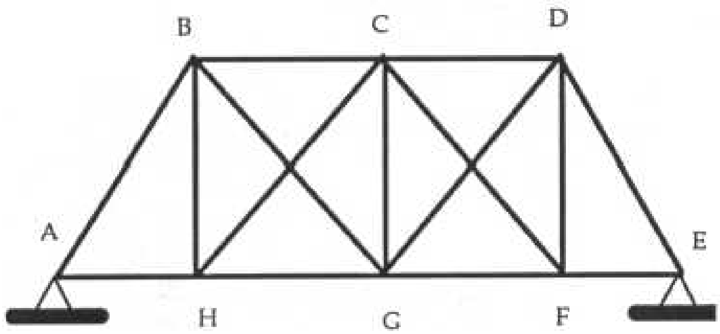
To compute the displacement of point G, we select the point, then select Analysis/Point Deflection from the menu.

To find the stress in the bar AB, we select the line then select Analysis/Stress from the menu.

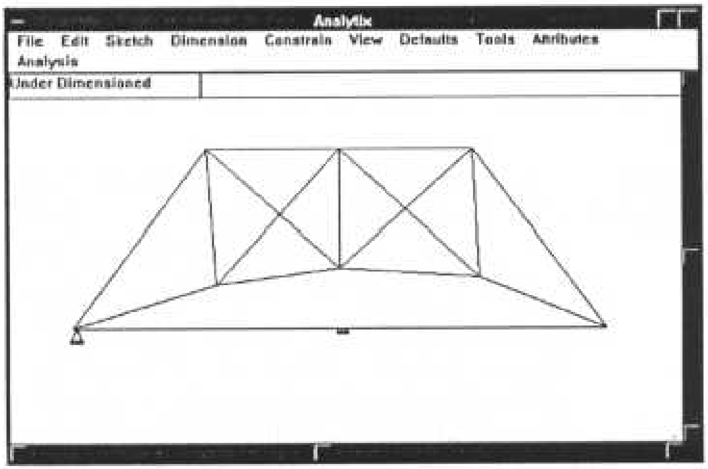


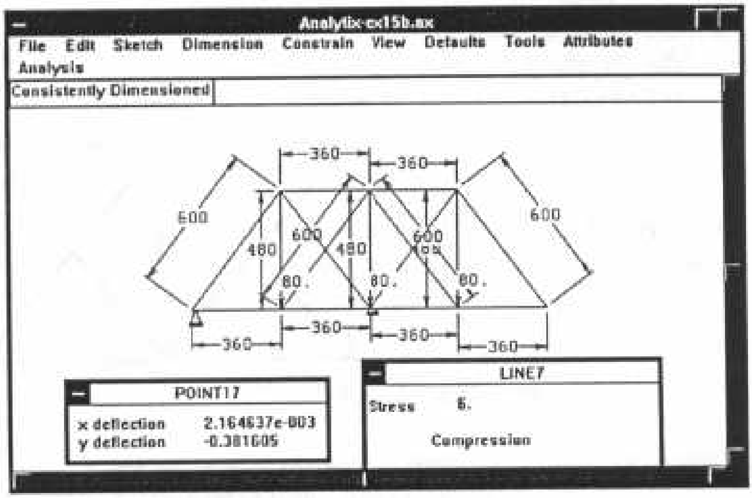
Variation

We now repeat the analysis for the situation where point E is fixed rather than free to roll. To model this situation, we create a fictitious bar between A and E and give it a large cross sectional area: say 1000 square inches. An exact model would have a bar with infinite cross sectional area, however a bar which is considerably thicker than others in the picture will give sufficient accuracy.

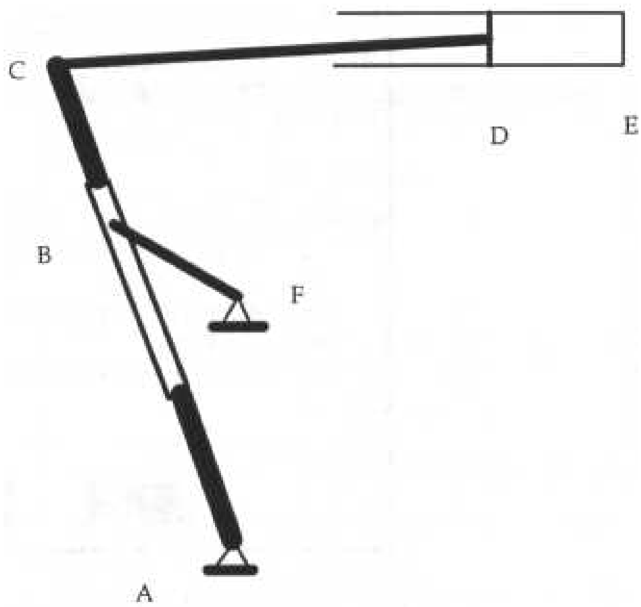


Reanalyzing the model shows that the deflection of point G is different, however the stress in bar AB is approximately the same with this new attachment.




Analytix-ex15b.ax
File Edit Sketch Dimension Constrain View Defaults Tools Attributes
Analysis
Consistently Dimensioned
360
360
600
600
600
600
480
480
80.
80.
80.
360
360
360
360
POINT17
x deflection 2.164637e-003
y deflection -0.381605
LINE7
Stress 6.
Compression

Example 15: Bending Moment & Shear Force

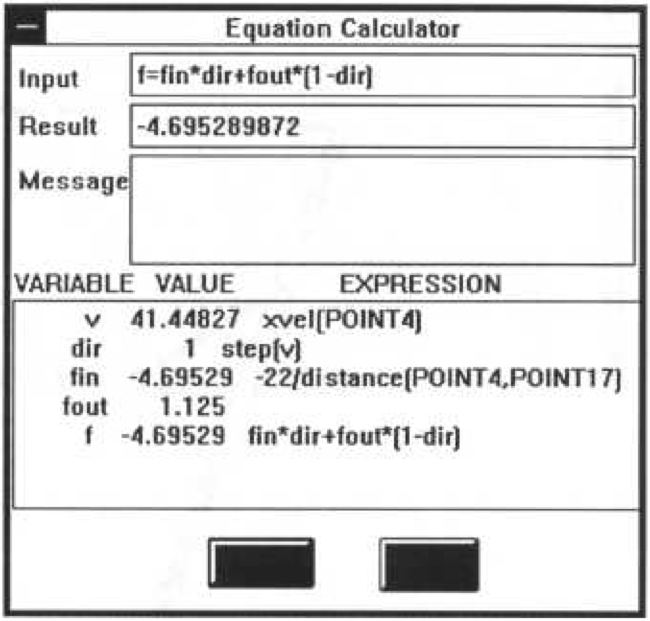


In this example we look at the bending moment and shear force at point B in the member AC during a cycle of the mechanism.

During the compression phase of the cycle, the piston at D experiences an opposing force of 22/ I DE I lb. During the expansion phase it experiences an opposing force of 1.125 lb. The piston weighs 1.251b. AC is 7"; BF is 1"; CD is 5". F is 3" above A; E is 4" above and 8" to the right of F.

We wish to look at two conditions: where BF is rotating at 30 rpm, and where BF is rotating at 300rpm.

The model



We define a variable f to represent the force on the piston. f is built up as follows:

v is the x-component of the velocity of the piston.

dir = step(v) (dir =1 if the piston is moving to the right; dir = 0 if the piston is moving to the left).

fin is the force in the compression phase.

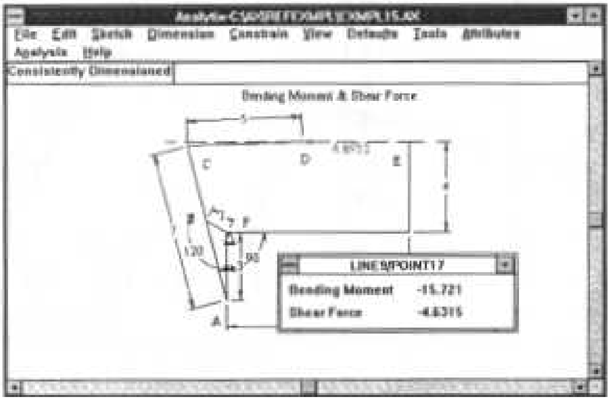
fout is the force in the expansion phase.

f = fin*dir + fout*(1-dir).

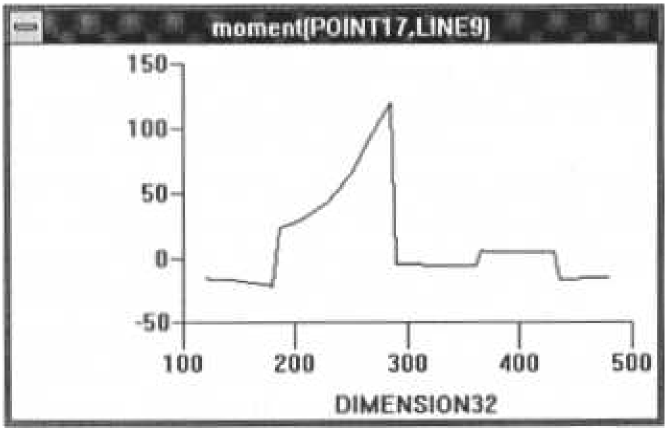
We draw the mechanism, set the mass of the piston, apply a force with x-component f and y-component 0 to the piston, and set an angular velocity of 3.14 rad/s to the driving angle. (We start with the 30 rpm example).

The Solution

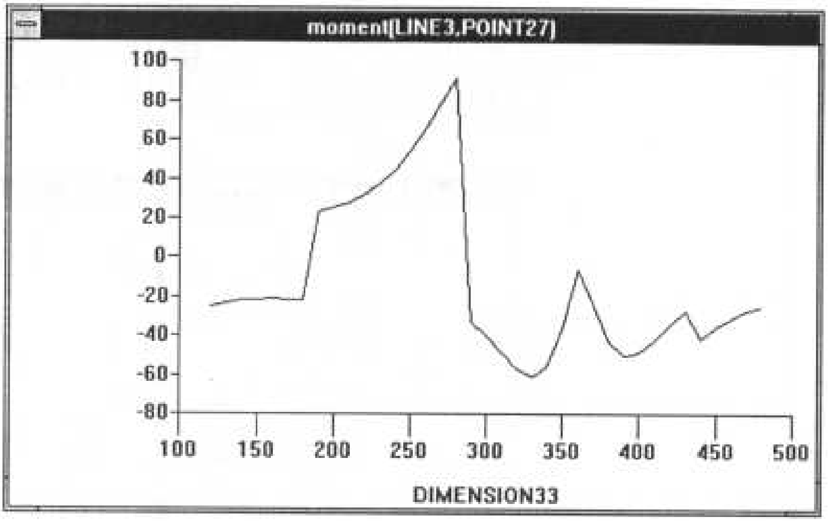
To derive the bending moment at B on line AC, we select the line and, holding down the *Shift* key select the point. Then we select Analysis/Shear/Bending Moment from the menu.



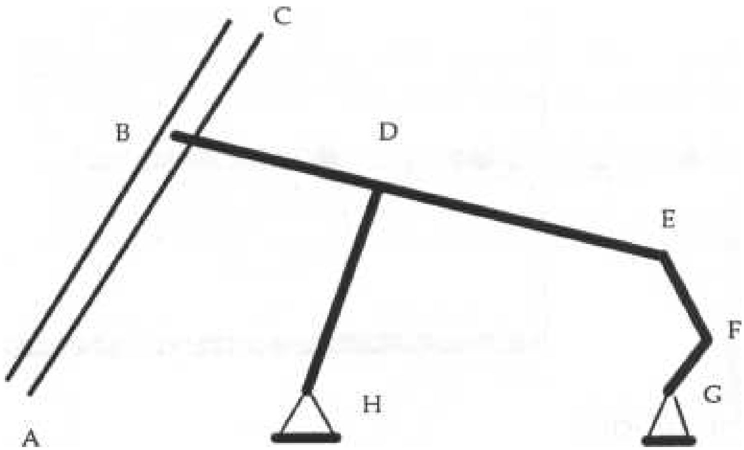
We can obtain a graph of the bending moment as the mechanism turns by graphing moment(POINT17,LINE9) against crank angle.



Changing the velocity of the crank angle to 31.4 lets us look at the 300 rpm case:



Example 16: Creating a Simulation

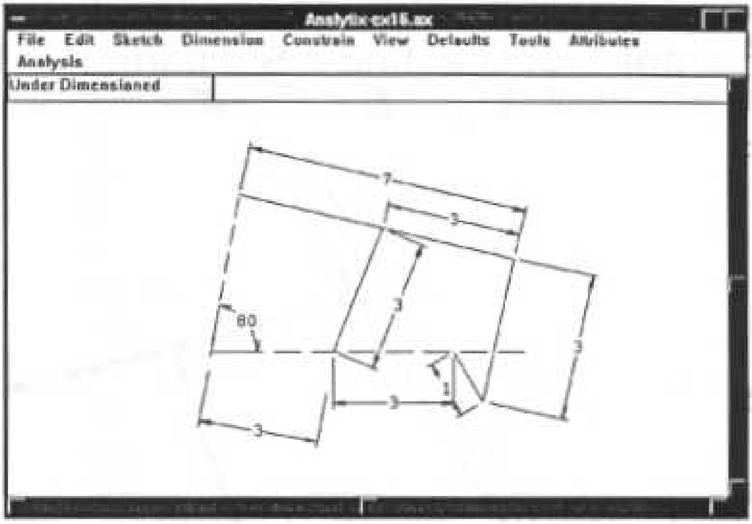


The problem

Some mechanisms cannot be modelled directly in Analytix. This occurs when the dimensions which are appropriate to specify the mechanism do not allow Analytix to construct the mechanism. In this example we explore the techniques which may be used to circumvent these limitations.

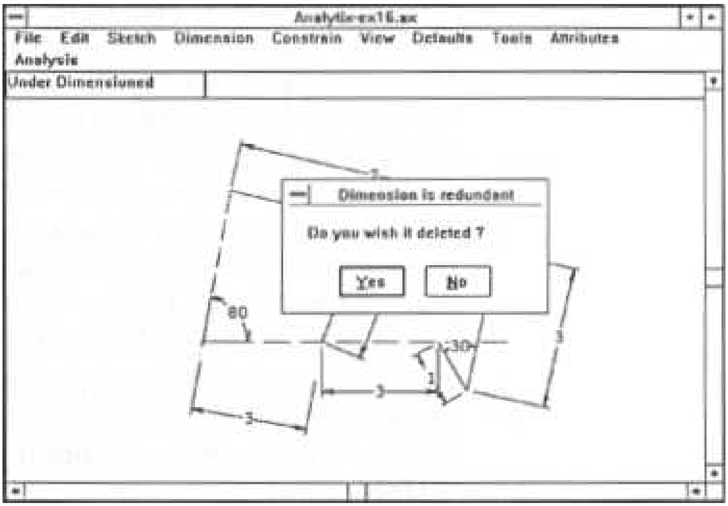
As an example we do a kinematic analysis of the above mechanism, where FG is the crank and B slides in the slot AC. Dimensions of the mechanism may be read off the Analytix drawing below.

In particular, we wish to graph the velocity and acceleration of B as the crank angle moves from 60 degrees to 150 degrees at a constant velocity of 10 rad/sec.



The Model:

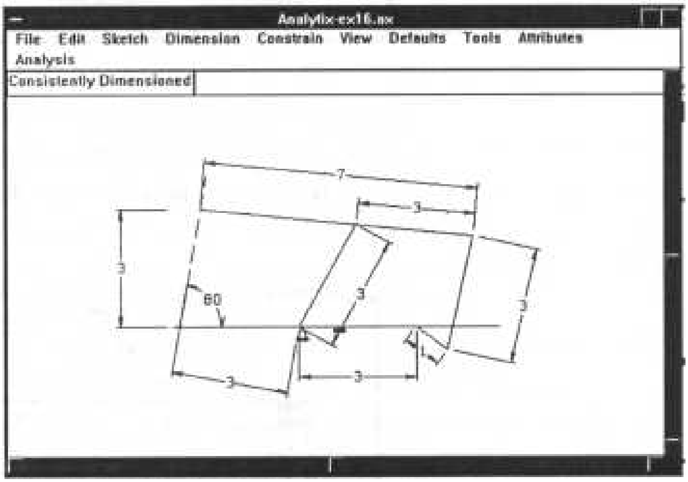
We wish to add the driving angle between the horizontal construction line and the crank to the above drawing. However Analytix responds by telling us that this angle is redundant. This means



that, using Constructive Variational Geometry, Analytix is unable to solve the geometric problem as posed.

If, however, we specify the height of the end effector rather than the crank angle, then Analytix can solve the geometry:

The problem is, we do not have any control over the crank angle, which is really the input parameter for

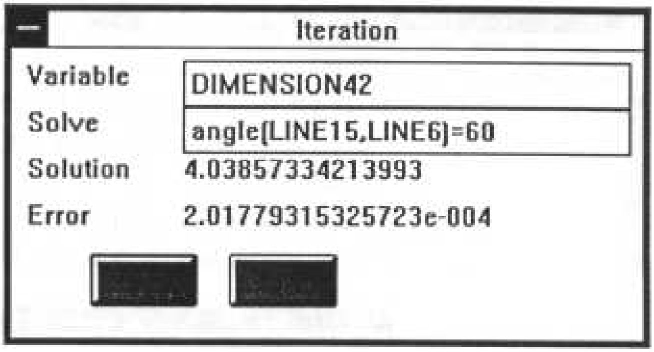


our motion.

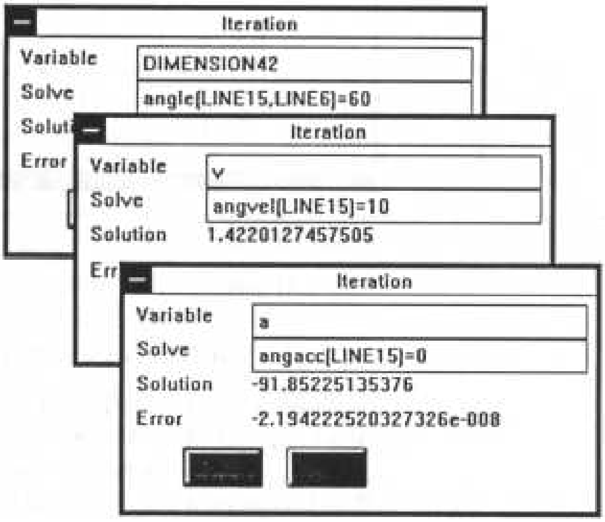
We can use the iteration tool, however to find the value for the height which makes the crank angle equal to a desired amount: say 60 degrees.

This gives us the appropriate geometry for a crank angle of 60 degrees. Now to get a correct kinematic model, we need to carry out two further steps: we need to iterate on the velocity and acceleration of the height dimension so that the angular velocity and angular acceleration of the crank have the appropriate values.

Let's assume we wish to drive the crank at a constant 3 radians per second. First we give a variable



velocity v and acceleration a to the height dimension. Then we create two more iteration boxes, one to find an appropriate value for v, another to find a value for a.



We now have an accurate kinematic model of this instant in the motion of the mechanism. If we get information on the end effector, we will see its velocity and acceleration for this instant in the motion.

To graph the velocity and acceleration of the end effector over a range of motion, we need to collect a sequence of instantaneous pictures of the mechanism in motion. We do this by creating a "new simulation".

A simulation collects a sequence of values for one or more dimensions and their velocities and accelerations. You first select the dimension(s) which you will vary (in this case the height of the end effector), then select Edit/New Simulation.

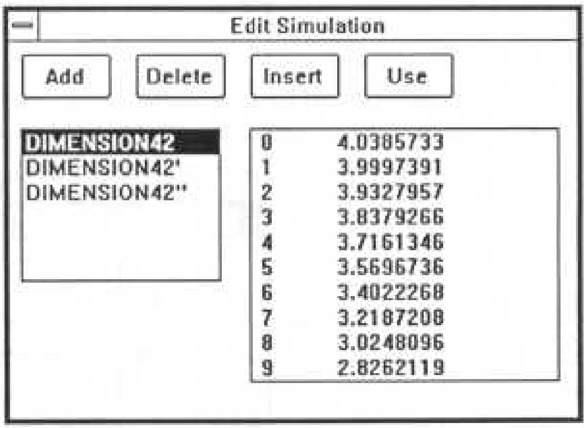
You are asked for the maximum size of the simulation. Enter 12.

You will then see the Edit Simulation dialog box.

To create your simulation, keep on screen the three Iteration boxes created above. Ensure you have the correct model for a crank angle of 60 degrees. Then press Add in the Edit Simulation Box.

The first instant of the simulation has crank angle 60.

Now change the angle in the first iteration box to 70; then solve each box in turn: first for the angle, then for the angular velocity, then for the angular acceleration.

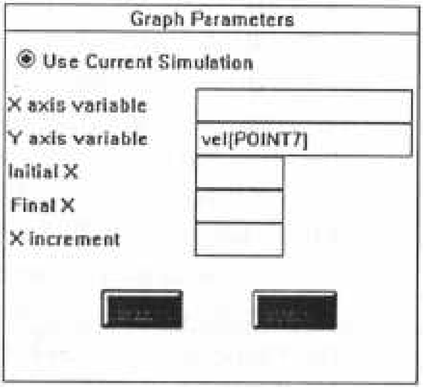


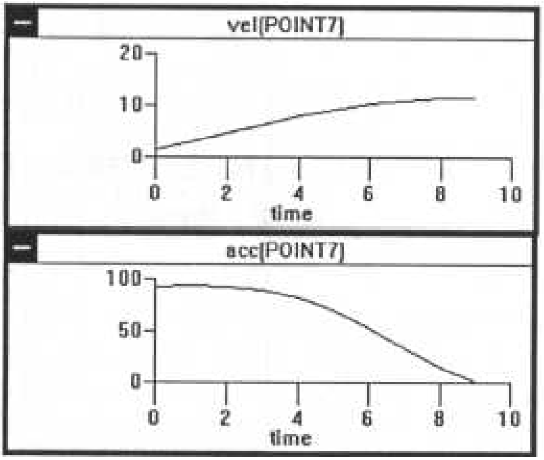
Now press Add in the Simulation box.

Now repeat the procedure for an angle of 80 degrees, 90 degrees and so on to 150 degrees.

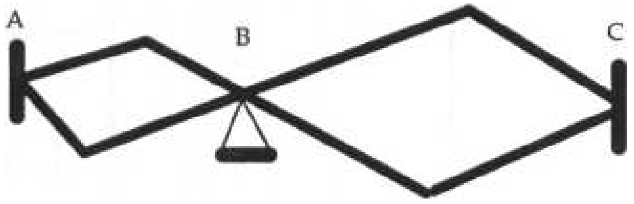
Now, to graph the velocity of the end effector, select Tools/Graph. You will see the graph dialog box. Select Use current simulation, then enter the expression to be graphed.

Use Current Simulation is also an option on the Table, Animate and Envelope tools.





Example 17: A Pantograph



The problem:

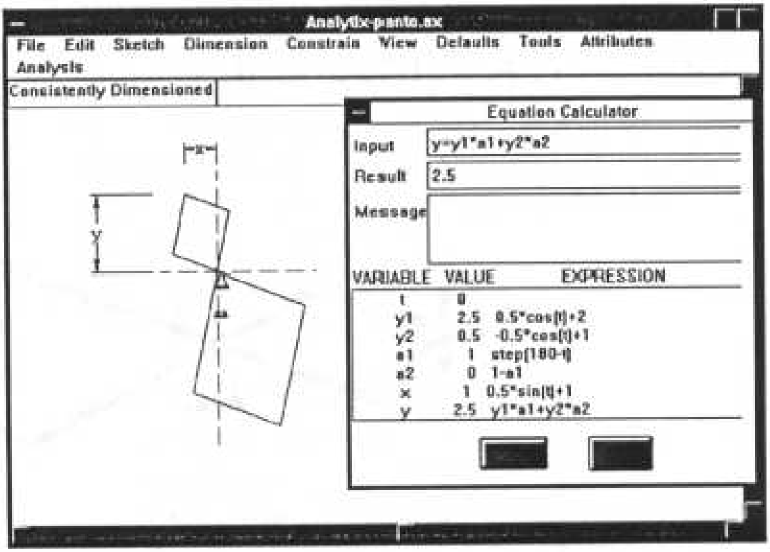
A pantograph is fixed at B and a pen at C generates a scaled inverted replica of the curve traced by stylus A.

In this example we model the pantograph as A traces a simple parametric curve depicting a letter

The model:

We draw construction lines to represent the axes, and a point at the intersection of the construction lines. We then draw the pantograph and specify that the point at the intersection of the construction lines lies on each of the intersecting bars of the pantograph.

The pantograph should be dimensioned so that the two parallelograms are similar. The relative sizes of the parallelograms determines the scaling factor of the device.



The location of point A is specified by its distance from the horizontal axis and its distance from the vertical axes.

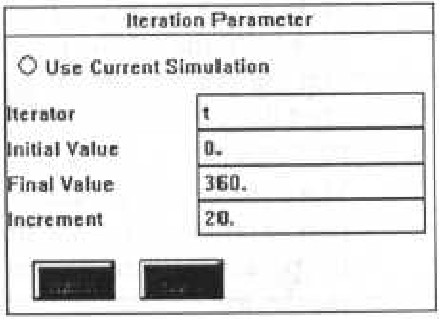
We specify this position by the parametric curve:

$x = -(1+0.5\sin(t))$

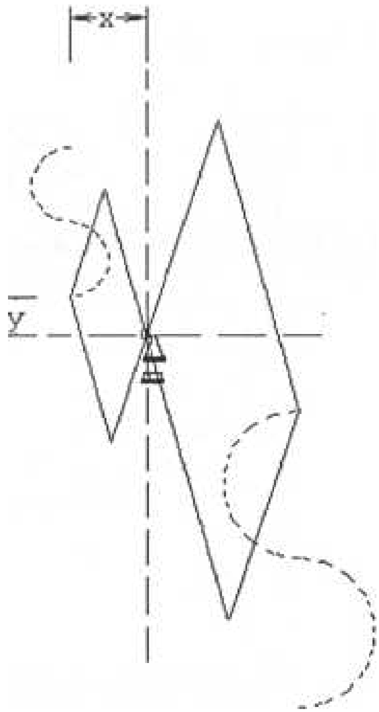
$y = 2 + 0.5\cos(t) \quad 0<t<180$

$y = 1 - 0.5\cos(t) \quad 180<t<360$

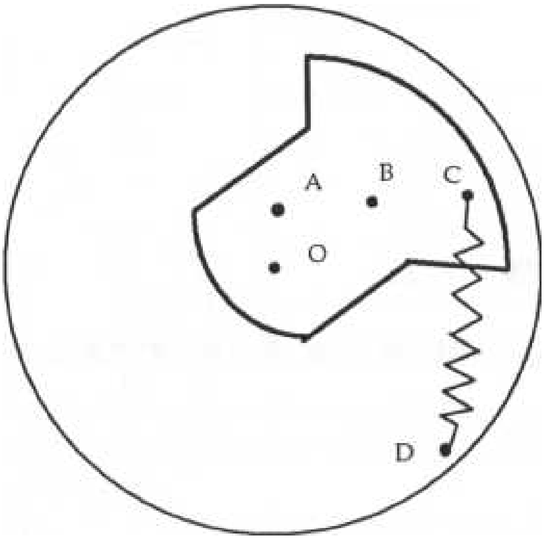
We can view the operation of the pantograph by animating with t as the variable over the range 0 to 360 in steps of 20.



Select both points A and C then select Tools/Trace from the menu to see the curve which the stylus follows and the magnified curve traced out by the pen.



Example 18: A Governor



The problem:

An engine governor is mounted on a flywheel rotating clockwise about O. It consists of an eccentric mass with center of gravity at B and weight O.llb pivoted on the flywheel at A.

A spring with stiffness O.llb/in and free length 2 inches is mounted on the flywheel at D and on the eccentric mass at C.

OA is 0.5", A,B and C are collinear and AB is 1" and BC is 1". OD is 3" and angle AOD is 135 degrees.

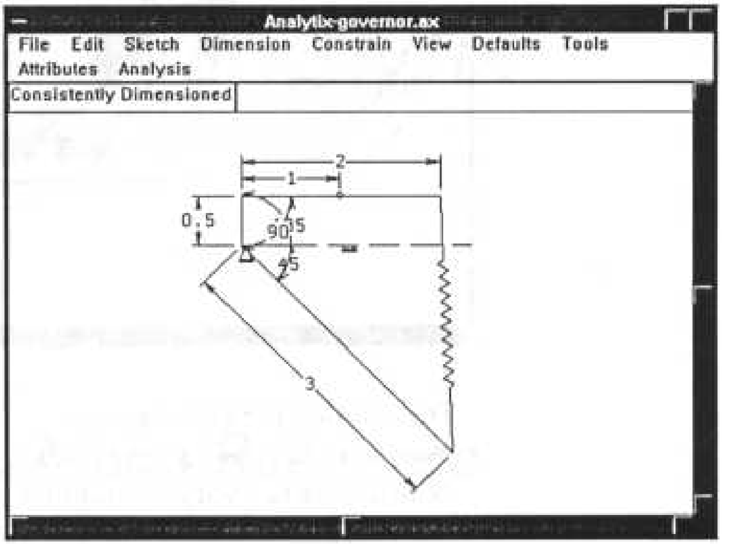
We wish to determine the angle OAC when the flywheel is rotating at 25 rad/sec.

The model

We create a fixed horizontal construction line as the background against which the flywheel will rotate. We create lines to represent OA, AC and OD, and a point on AC to represent B.

We use Defaults/Dynamic Defaults to set our units to ips.

We set the angle between OD and the horizontal to be (arbitrarily) 45 degrees, and give this angle a velocity of 25.



We also set the angle OAC to an initial value of 90 degrees.

To attach a spring to CD, we select both points then use Analysis/Add Actuator.

The solution:

To find the steady state value for the angle OAC, we use the iteration tool. The angle OAC is free to move in the device, it will therefore only be in equilibrium if the torque transmitted by the angle is 0. Hence we need to find the value of the angle for which the reaction torque in the angle is 0.

The solution is 88.85 degrees.

If we wish to create a graph of angle versus angular velocity, we can vary the angular velocity, repeat the iteration performed above and capture the results in a simulation.

Below we see a graph for governor angle as a function of angular velocity

Analytix

design the future

Menu Reference

Menu Reference

In this section, we systematically examine the different menu options available in Analytix.

First we look at each of the options available on the main menu and make general statements about the different menu selections to be found in the corresponding drop down menu.

Secondly, we examine each individual menu option in turn and describe the corresponding functions thus invoked.

Main Menu Options

System

The System menu contains the standard Microsoft Windows options for sizing, closing and moving the main Analytix Window.

File

The File menu contains options to let you create a New file, Open an existing one, save the current drawing as a file, and Print the current drawing. It also contains functions to read and write DXF files.

Edit

The Edit menu lets you Select portions of your drawing. It also lets you Cut, Copy or Paste selected portions of the drawing or Snap bitmaps from the screen.

Sketch

The Sketch menu contains all the commands to let you sketch drawing entities: Points, Lines, Arcs, Fillets, Circles, and Construction Lines are all here.

Dimension

The Dimension menu is where you can add dimensions to your sketch in order to convert it into a scale drawing.

Constrain

The Constrain menu contains options which allow you to specify which point and line of the drawing will stay fixed in any motion or statics problem. Further menu options allow you to specify line segments in the sketch as being portions of the same lines, and specify circles to be concentric.

View

The View menu lets you move or rotate selected portions of the drawing, it lets you zoom in or out, it lets you blank or unblank portions of the drawing and do level management.

Defaults

The Defaults menu lets you set default pen colors and styles, set default unit types, set default tolerances and specify whether tolerance analysis is to be statistical or absolute.

Tools

The Tools menu contains functions which let you animate your drawing, create an Envelope of it, or Trace the curve followed by a given point. It also has tools for creating Graphs and Tables of values of interest, a Calculator and Equation Solver.

Attributes

The Attributes menu lets you view the various attributes of all the drawing entities and dimensions in the drawing. It lets you change whichever attributes are appropriate to change. There is also a function which lets you measure **distances** and angles from the drawing.

Analysis

The Analysis menu lets you add loads to the drawing, derive reaction forces and tolerance zones.

Help

The Help menu shows you step-by-step procedures for every Analytix option.

System

The System menu contains commands which are included in all Microsoft Windows applications for moving sizing and closing the window.

It is activated by clicking on the small box in the top left hand comer of the Analytix window, or by pressing [ALT] + [SPACEBAR].

System / Restore

Restores a window to its original size, either by expanding it from Iconic (or Minimized) form or reducing it from full screen (or Maximized) form.

This option can be invoked by clicking on the double arrow box at the top right of your Analytix window, or from the keyboard by pressing [ALT] + [F5].

System / Minimise

This causes Analytix to go Iconic. The whole window is reduced to an Icon at the bottom of the screen.

This option may alternatively be invoked by clicking on the down arrow at the top right hand corner of the Analytix window, or by pressing [ALT] + [F9].

System / Maximise

This option causes the Analytix window to cover the entire screen. This is convenient if you are working solely in Analytix, but less convenient for switching between windows.

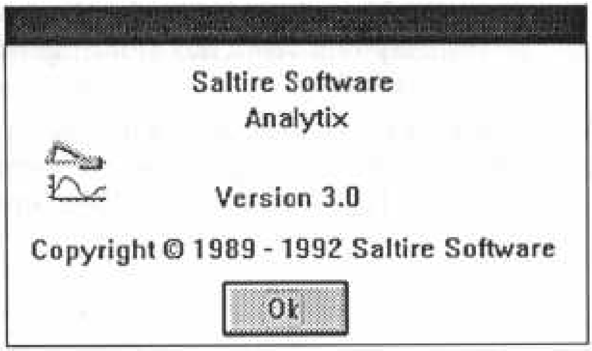
This option may be invoked by clicking on the up arrow at the top right hand corner of the Analytix window, or by pressing [ALT] + [F10].

System / Close

This closes the Analytix application. The option may alternatively be invoked by double clicking in the System Menu Box at the top left corner of the window.

System / About

This option brings up the Analytix copyright notice.



File

The file menu contains options which perform a number of disk and printer / plotter related tasks: reading and writing drawings to disk, reading and writing DXF files for communication with CAD systems, and plotting.

File / New

Creates a new file with no name and no contents.

Erases the current drawing.

If your current file has not been saved, Analytix will ask you whether you wish to save it before erasing it.

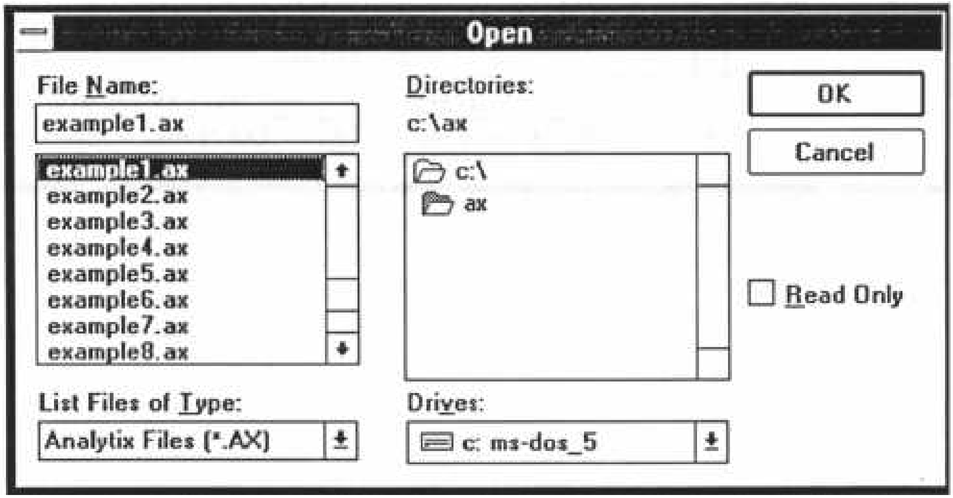
File / Open

This menu option allows you to open a previously saved Analytix file.

Analytix presents you with the File dialog box. It contains a file list box which will initially be filled with all the files with the ax extension (the default Analytix extension). The dialog box also contains a file entry area, where the name of the file to be loaded may be typed in.

To select one of the files in the directory box, click on the file name. Notice that this name will be echoed in the filename box.

The scroll bar on the side of the file list box may be used to scroll through the files.



Alternatively the file name may be typed directly into the filename box.

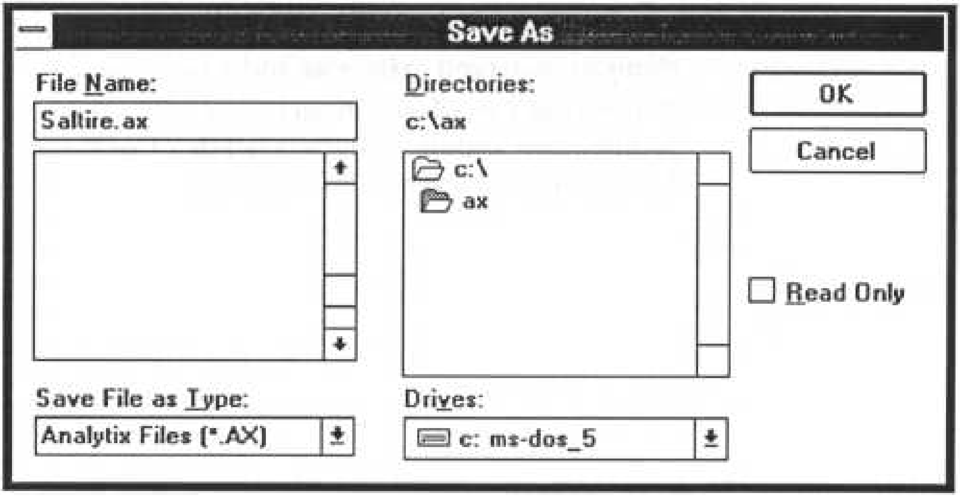
When the correct file name has been entered, click on the OK button, or press the Enter key.

File / Save

This option saves the current drawing in a file with the current name. If the current drawing is untitled, you will be prompted to give the file a name in a similar way to the Save as... option below.

File / Save as...

This option prompts you to give the name of the file in which the current design is to be saved. A file is then created with the given name and the extension '.ax' if no alternative file extension is given.



File / Plot

Generates a plot of your drawing on the default printer or plotter.

To change which printer is selected as the default printer, you can use the Microsoft Windows Control Panel. This is invoked by double clicking on the Control Panel icon in the Main window.

The Printers icon in the Control Panel allows you to change the default printer or add a new one.

You may change the configuration of the current printer from within Analytix; see File / Configure Printer option on the following page.

File / Configure Printer...

This option brings up a dialog box which lets you configure the current default printer. This lets you choose for example, between landscape and portrait presentation, and set whatever parameters your printer has which may be altered.

The exact form of this dialog box depends on what specific type of printer is installed.

This dialog box is the same one as appears in the Printers Setup utility of the Windows Control Panel.

File / DXF Out

This option lets you output geometry as a DXF file. This facilitates transfer between Analytix and other CAD programs.

A filename box will appear into which you should type the name of the DXF file. The default extension .DXF will be added if no file extension is given.

In the current release of Analytix, only geometry entities are saved in DXF files. Dimensions are not saved.

File / DXF In

This option lets you read in DXF files created in other CAD systems.

When you select this option, you will see the same file selection box which appears with the File / Open menu option. This time files with the extension .DXF will be preselected in the File List box.

To read in a DXF file, select or type in the name and Click on the Open button.

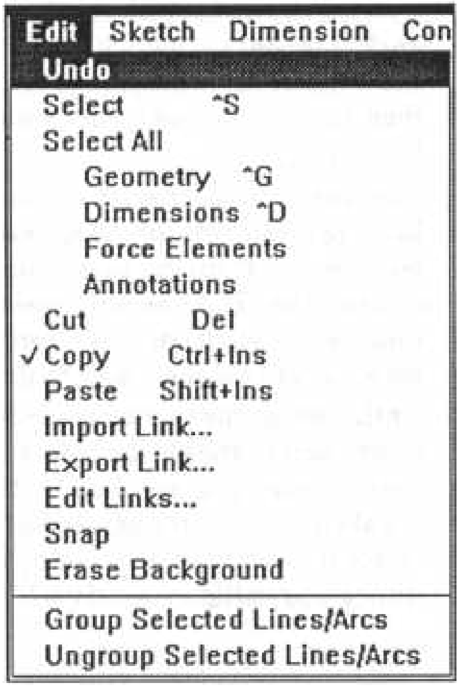
The current release of Analytix is only able to read geometric information from DXF files, and not dimensions.

In DXF files dimensions are regarded as ornaments and the geometry is defined by the drawing. In Analytix the dimensions are the things which define the geometry. This difference in outlook explains the fact that the DXF files are not rich enough to preserve a full dimension driven geometry description.

If you are importing a correctly sized part from a CAD system, and do not care how it is to be dimensioned, you can have Analytix automatically dimension the part using the Dimension / Automatic option.

Note that if your DXF file contains more than 350 entities, you will not be able to read it into Analytix. It is best to keep the size of Analytix files moderate to avoid excessive degradation of the system's performance.

Edit



The Edit menu contains the Selection options, which are used to select which entities certain operations will be performed upon.

The Edit menu also contains the standard Cut, Copy, Paste operations , the Undo function, Erase Background and the Snap function which allows you to take a "snapshot" of your screen to be included in some other document, perhaps a word processor, using the Microsoft Windows Clipboard. The Group Selected... allows Analytix to display mass properties of the selected profile.

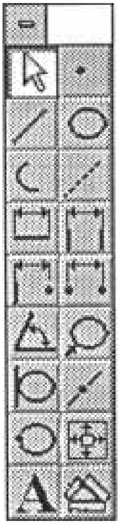
Edit / Undo

When a sketch is consistently dimensioned, Analytix automatically converts this dimensioned sketch into a scaled drawing. If the sketch and the dimensions are too far out of line with each other, the results of this process can be a little confusing. It can turn out that the scaled drawing does not look like the intended picture.

The root cause of the dimensioned drawing not looking like it was intended is that the sketch is too far removed from the part defined by the dimensions. This can either be because one or more dimensions were entered wrongly, or because the sketch is not close enough to the intended part.

At this stage, the Undo option is available and will return you to the sketch before Analytix redrew it. You can now remedy the situation either by altering the sketch (using the Move, Rotate, and Select-then-drag options) or by altering one or more dimension values.

Edit / Select



This option is used to select entities in the drawing. Selected entities can then have one of a number of operations performed upon them. They can be Cut, Copied, Blanked, Moved, Rotated, and, depending on the type of the selected entities, a variety of more specialized functions may be performed.

Many of the menu items are "grayed out" and unpickable unless some entities (sometimes of a specific type or combination of types) are selected. Examples are the Cut and Copy options in the Edit menu, which become active only when some entities are currently selected. A more complicated example is the Attributes / Measure option which is only active when either a pair of points, a pair of lines, or a line and a point are selected.

To enter the select mode click on the arrow icon in the toolbox

Keyboard Shortcut:

Press [Control] + S

Mouse Shortcut:

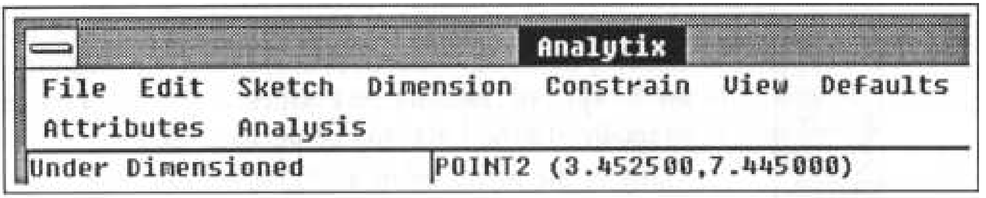
Click on the right mouse button. This is the only use for the right mouse button in the system.

Selecting a Single Entity

To select a single entity, position the cursor over that entity and click with the left mouse button.

If there are too many entities in the same region you can use the View / Zoom commands to zoom in on the region of interest. You should thus be able to spread the entities out sufficiently to discriminate the entity which you wish to select.

When an entity is selected, its name and some information about it appears in the Information box at the upper right of your Analytix screen.



Except for points, selected entities are drawn with the current highlight pen style and color. By default this is a solid red on color systems and a broken black line on black and white systems. These defaults may be changed using the Defaults / Default Pens menu option.

If more information about the entity is required, use the Attributes I Info menu option.

Unless you hold down the shift key (see Selecting Multiple Entities below), when you select an entity by clicking over it, whatever was currently selected becomes deselected and the entity which you clicked on becomes the only selected entity.

Note that only entities which are visible on the screen may be selected in this way. Blanked entities for example are not selectable.

Selecting Multiple Entities

To select multiple entities, hold down the [SHIFT] key when you click on the entity to be selected. Then the previously selected entities are not unselected and the entity you click on is added to their number.

Information about the latest selected entity still appears in the Information box at the upper right of the Analytix window.

Selecting Points

To select a point click on that point. Points are not highlighted when picked, but the point information appears in the Information box.

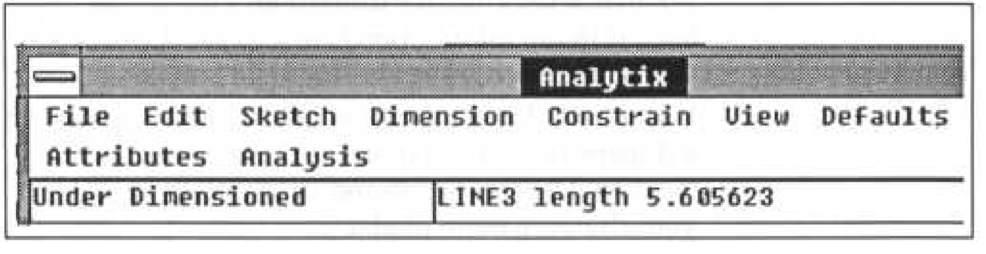
If you hold down the mouse button after you have clicked on a point, you can drag that sketched point. This is referred to as "Select-and-drag", and is a convenient way to alter the shape of your sketch.

A more sophisticated way to alter your sketch is by selecting a number of entities then using the View / Rotate and View / Move menu options to rotate and move the selected entities.

Selecting Lines

To select a line, click on that line away from the endpoints. If you are too close to one of the endpoints, that point will be selected rather than the line.

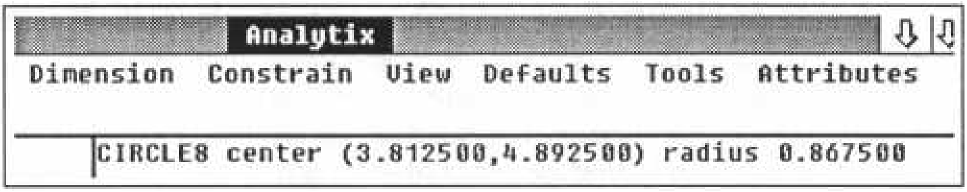
When it is selected, the line's name, and its length will appear in the Information Box. It will also be drawn in the highlight pen style / color.



Selecting Arcs and Circles

Select an arc or circle by clicking on its circumference (away from endpoints in the case of an arc). If you click on the center of a circle you will select only that point.

When it is selected, the circle's name, center and radius will appear in the Information box. The circle will be drawn in the highlight pen style / color.



Note that for a variety of purposes, selecting a circle or arc is equivalent to selecting the center of the circle. For example if you select a circle and a line and then pick the Attributes / Measure menu option, you will be given the distance of the center of the circle from the line. This is the same as if you had selected the center point and the line.

Selecting a Dimension

Select a dimension by clicking on or near the text of the dimension.

When it is selected, the dimension's name will appear in the Information box, and the dimension will be redrawn using the highlight pen color / style.

If you hold down the mouse button when you select a dimension you can drag the dimension to a new location. Use this Select-and-drag option to make your drawing more readable.

To change the value of a selected dimension use the Attributes / Info menu option.

Selecting a Force / Torque

Select a force or torque by clicking on or near the force or torque value.

The force arrow or torque arc will be highlighted.

Edit / Select All

This option selects all the entities in the current drawing. Both blanked and unblanked entities are selected.

For example Select All followed by Cut results in the entire drawing being removed and put on the clipboard.

Edit / Select All Geometry

This option selects all the points, lines, arcs and circles in the current drawing. Dimensions are not selected. Both blanked and unblanked geometrical entities are selected.

Keyboard Shortcut:

Press [Control] + G.

Edit / Select All Dimensions

This option selects all the dimensions in the current figure. Geometric entities are not selected. Both blanked and unblanked dimensions are selected.

Keyboard Shortcut:

Press [Control] + D.

Edit / Select All Force Elements

This option selects all applied forces and torques in the current drawing. Both blanked and unblanked force elements are selected.

Edit / Select All Annotations

This option selects all annotations made to the current drawing. Both blanked and unblanked annotation entries are selected.

Edit / Cut

This option Cuts the currently selected entities and any dependent entities from the drawing. These entities are then pasted onto the clipboard.

What is removed

All the currently selected entities are removed from the drawing. In addition, any dimensions, applied torques or forces which refer to any of the selected entities are removed. For example, if we have a triangle dimensioned by the length of its three sides, and we cut one of the lines, the line length dimension referring to that line will also disappear.

Points are of two types:

l - Explicit - these are drawn using the Sketch / Point menu option and are drawn as tiny circles.

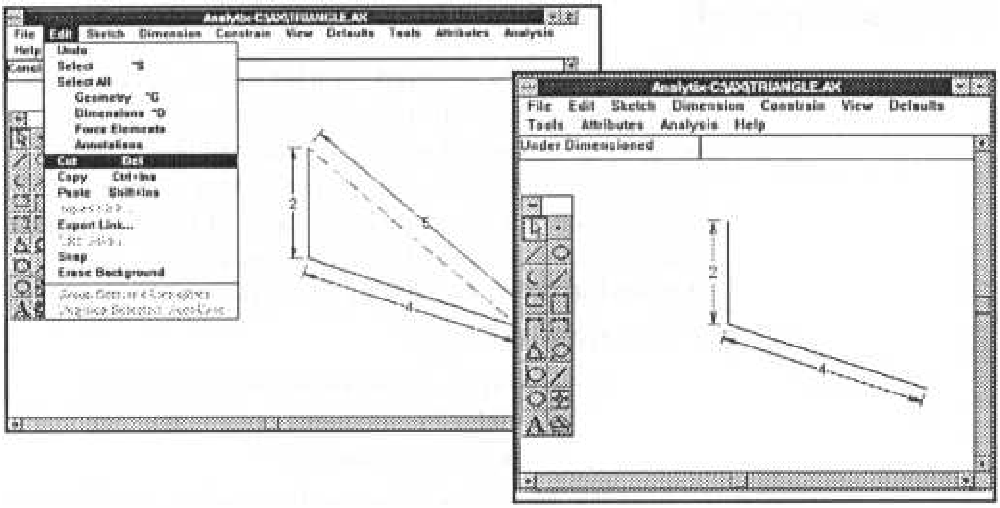
2 - Implicit - these are line and arc endpoints and arc and circle centers.

Explicit points are cut only when selected. Implicit points are cut when all the lines arcs and circles which use these points are cut. For example, if one line of our triangle is cut, no implicit points are removed, as both end points of the selected line are also endpoints of other unselected lines. If we were to cut two lines, however, the shared endpoint would also be cut.

What appears on the clipboard

When you cut a selection of entities from a drawing, the clipboard receives a description of those entities in Analytix format, plus any entities dependent solely on those cut, minus any entities dependent on other entities which were not cut.

Any entities which are dependent on entities which were not Cut are not placed on the clipboard. For example if a line length dimension is cut but the line which it refers to is not cut, the dimension is indeed removed, but it is not placed on the clipboard.



The Analytix format description on the clipboard has all the information needed to Paste these entities into another Analytix drawing. However this format is not readable by any other programs.

If you use Copy rather than Cut, both an Analytix format description and a Metafile picture is stored on the clipboard. The Metafile picture format is understood by a number of Windows applications.

The most widely used way of communicating graphics information with other Windows applications is by pasting a bitmap onto the Clipboard. This is done in Analytix using the Edit / Snap menu option, described later in this section.

Numerical values may be copied in a form suitable for pasting into the Excel spreadsheet or into a

Word Processor document using the Copy option in the Table window's menu. A table is generated using the Tools / Table menu option.

When a line, circle, or arc is put on the clipboard, endpoints and centers for those entities are also put on the clipboard, regardless of whether those points were removed from the drawing or not. This means that Cut followed by Paste is not necessarily a null operation, as additional implicit points may be generated.

Edit / Copy

This option copies all currently selected entities to the clipboard. Any entities which depend on unselected entities are not copied.

The selected entities are not removed from the drawing.

What appears on the Clipboard

All the selected entities appear on the clipboard drawn in Metafile Picture format. This format may be pasted into a number of different Microsoft Windows applications. The Metafile Picture format keeps a description of a drawing as a sequence of drawing commands. This can be preferable to a bitmap as it takes up less memory, and can be redisplayed to whatever resolution is available, which may be considerably greater than the screen where the picture was originally drawn (this is often the case, for example with laser printers).

Copy also puts an Analytix format description of the selected entities on the Clipboard. This may later be pasted into another Analytix window or on the same Analytix window to create a duplicate of the selected sub-drawing.

Any entities which are dependent on entities which were not copied are not placed on the clipboard. For example if a line length dimension is copied but the line which it refers to is not , the dimension is not placed on the clipboard.

When a line, circle, or arc is put on the clipboard, endpoints and centers for those entities are also put on the clipboard

To copy bitmap pictures to the Clipboard use the Edit / Snap menu option described later in this section.

Numerical values may be copied in a form suitable for pasting into the Excel spreadsheet or into a Word Processor document using the Copy option in the Table window's menu. A table is generated using the Tools / Table menu option.

Edit / Paste

If you have previously Cut or Copied an Analytix drawing to the clipboard, this option allows you to Paste that drawing onto your current drawing.

Paste is "Grayed out" unless the clipboard contains a drawing in Analytix format. This drawing may have been Cut or Copied from the current Analytix window, or from a separate Analytix window.

When you pick Paste, the entities in the clipboard will be added to your current drawing. The pasted entities will then become the currently selected entities. They will be positioned in the same coordinate position they were Cut or Copied from.

This can be a little confusing under certain circumstances:

1 - If the entities are pasted outside the current screen window - then you will not immediately see them. You can use Zoom max/min from the View menu to see the whole picture.

2 - If the entities have just been Copied from the same window - then the new copy will be pasted on top of the old ones. You can use View / Move to move the new copy so you can see it.

It is important to note that Cutting an entity then Pasting it back in does not necessarily give you the same object you started with because of the additional entities which are added to the clipboard along with selected entities.

For example, if you cut one edge of a triangle, the line and a pair of end points are put on the clipboard. However the line's end points are not removed from the drawing as they are necessary to define the ends of the other lines of the triangle. When you Paste the line back in, both the line and its new endpoints are pasted. Thus the line is no longer attached to the rest of the triangle and may be Moved away from it. To reattach the line you would need to use the Same Point constraint.

A Copy & Paste Example

We give a brief example of copying and pasting. We draw a hole pattern, and create a second copy of the pattern using Copy and Paste. The steps are as follows:

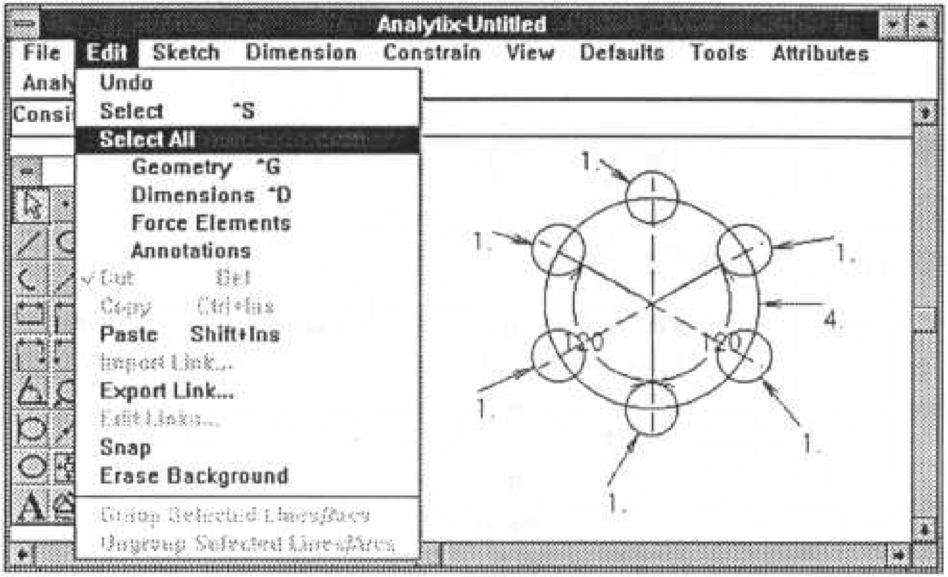
1 - Draw the hole pattern

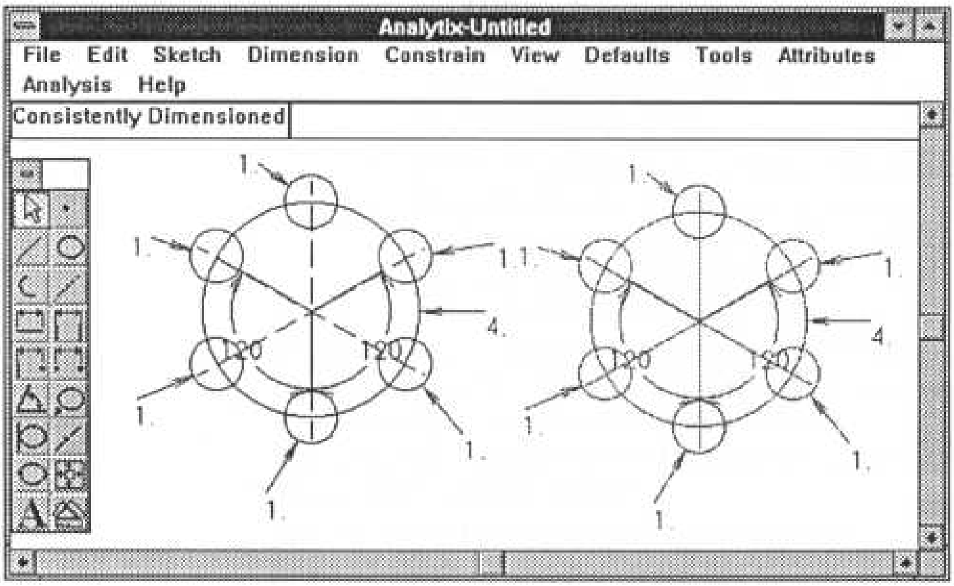
2 - Select All

3 - Copy

4 - Paste

5 - Move the copy to a new location


Analytix-Untitled
File Edit Sketch Dimension Constrain View Defaults Tools Attributes
Undo
Select ^S
Select All
Geometry ^G
Dimensions ^D
Force Elements
Annotations
Paste Shift+Ins
Export Link...
Snap
Erase Background
1.
4.
120
120


Analytix-Untitled
File Edit Sketch Dimension Constrain View Defaults Tools Attributes
Analysis Help
Consistently Dimensioned
1.
4.
120
120

Edit / Import Link...

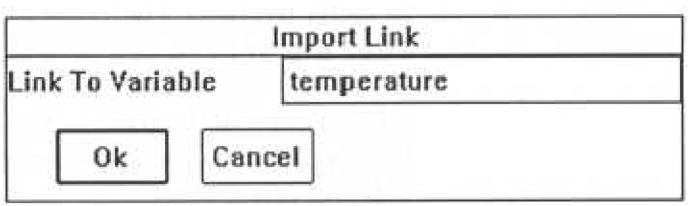
This option lets you set up a Dynamic Data Exchange (DDE) link which feeds data into Analytix from another program running under Windows (or from another copy of Analytix running under Windows).

Edit / Import Link is selectable only if there is a Link currently on the Clipboard. Hence, before selecting this option, you must ensure that a link is present on the clipboard. How this is done depends on which program you are linking to Analytix.

If you are creating a link from another copy of Analytix, you do this using Edit / Export Link.

If you are creating a link from Microsoft Excel, you use Edit / Copy.

When you select Import Link, you see the Import Link dialog box. Use this box to specify the name of the variable to which the imported values are to be given. This may be any legal variable name or the name of a dimension in our Analytix drawing.

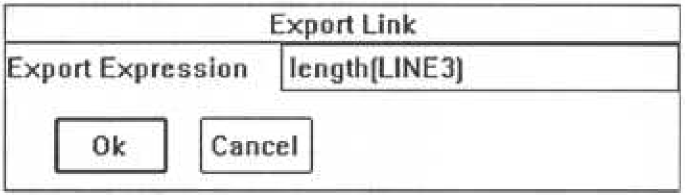


Once a DDE link has been imported into Analytix, any change in the exported value will be automatically reflected by a change in the variable to which it is linked in Analytix.

Edit / Export Link...

This option is used to export a Dynamic Data Exchange (DDE) link from Analytix to another program running under Windows (or to another copy of Analytix running under Windows).

When you select Edit / Export Link, you will see the Export Link dialog box.



You may enter into this box any legal Analytix expression. Examples of such expressions are:

12.5

x

x^2-sin(theta)

angle(LINE4,LINE6)

In order to create a working DDE link, after using File / Export Link. you then have to import the link from the clipboard into a second application (or second copy of Analytix). The command used for this depends on the application with which the link is being established.

If you are creating a link with a second copy of Analytix, you use Edit / Import Link for this.

In Microsoft Excel, you use Edit / Paste Link.

Once a link is established, whenever Analytix has reason to believe that the value of the linked expression may have changed, the updated value is sent to the other program, which in due course should recalculate and refresh its display based on this new information.

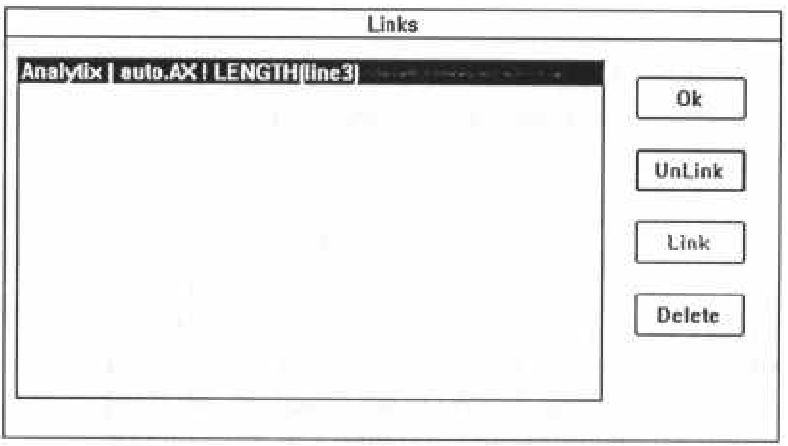
Edit / Edit Links...

This menu option lets you view the links which have been imported into Analytix. It also lets you temporarily or permanently disconnect the links.

When you select the option you see the Link Edit dialog box. This has a list of existing imported links, recorded in the following way:

Application | Topic ! Item

The application is typically the name of the program where the link initiated. The topic is typically the name of the current file being handled by that



program. The item is typically the name of the individual piece of information which has been linked. If the link is with another copy of Analytix, the item is the expression which was exported. If the link is with Excel, the item is a spreadsheet cell identifier.

When you import a link to Analytix from another Windows application (or from another copy of Analytix) whenever the exporting application believes that the value of the link may have changed, it sends a new value to Analytix. Analytix then automatically recomputes the current model based on this new information.

Sometimes it is convenient to suspend this behavior temporarily (for example to enable you to make a sequence of changes in the exporting application). You can do this by selecting the appropriate link in the link list box. Then press the Unlink button.

To reestablish the link, select the link in the link list box, then press the Link button.

To permanently close a link, select the appropriate link in the link list box. Then press the Delete button.

Edit / Snap

This menu option lets you transfer a bitmap picture (or snapshot) of part of your screen to the Clipboard.

You can then Paste this picture into another Windows application, perhaps a Word Processor.

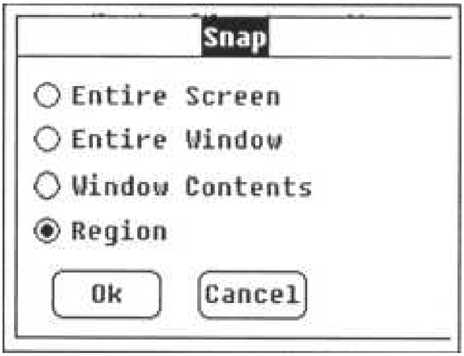
When you select Edit / Snap, you will be give the choice of snapping either

1 - The entire screen.

2 - The current Analytix window only

3 - The contents of the current Analytix window (without the menu bar and surrounding box.

4 - A region of your choosing.



If you select one of the first three options, the snapshot will be taken as soon as you click on Ok. If you select Region, you must first click on Ok, then specify the region to be snapped by pointing to its top left corner, depressing the mouse button and dragging to the bottom right corner of the required region. When you lift the mouse button, the bitmap will be pasted onto the Clipboard.

You can inspect the Clipboard using the Microsoft Windows Clipboard utility.

The Snap function converts Color bitmaps to black and white in order to save on memory.

If there is not enough memory to save your bitmap, you will be presented with a message to that effect.

Edit / Erase Background

Analytix has a special Background layer. This is regarded as a sheet of paper behind the model, which certain tools can draw lines and curves on. The Edit / Erase Background option erases this background layer.

The specific things which appear on this background layer are:

1 - Point traces constructed with the Tools / Trace menu option.

2 - Tolerance zones constructed with the Analysis / Tolerance Zone menu option.

Edit / Group Selected Lines / Arcs

This option lets you create a group out of a set of lines arcs and circles. The main use of this feature is to define a profile in order to compute area mass properties.

To create a group, first select all the lines and arcs which are to be included (holding down the Shift key to create a multiple entity selection). Then choose Edit / Group Selected Lines / Arcs.

The selected profile is now treated as a group. Whenever you select one of the constituent entities, you will find that the whole group will be selected.

To derive the area mass properties, you should select the group then select Attributes / Info. (A shortcut is to double click on the group.)

If you wish to select individual entities within the group, you must first ungroup it using the Edit / Ungroup Selected Lines / Arcs menu selection.

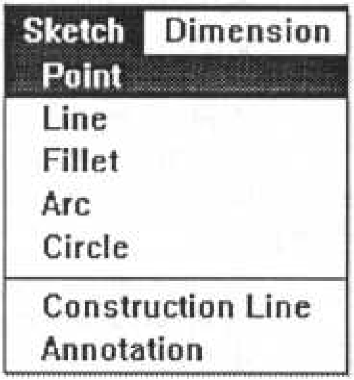
Edit / Ungroup Selected Lines / Arcs

Groups are created in Analytix using the Edit / Group... menu option. Once a group is established, its constituent members may not be selected individually, they can only be selected all at once.

The Edit / Ungroup... menu option allows you to break up a group in order that its individual members may again be selected.

To use this menu option, first select the group by clicking on one of its members. Now choose Edit / Ungroup....

Sketch



Create a drawing of an object in Analytix in two basic steps:

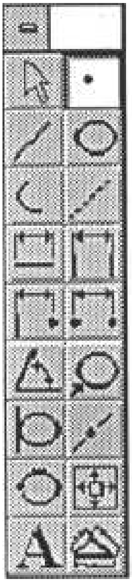
1 - Sketch the approximate shape of the object.

2 - Add dimensions to specify the exact geometry of the object.

The commands in the Sketch menu let you draw lines, arcs, circles, points fillets and construction lines.

All the Sketch commands in Analytix are modes. This means that once you have picked, for example, Sketch / Line, you can continue sketching lines until you make another menu choice.

Sketch / Point



There are two types of points in an Analytix drawing

1 - Implicit - these are created as a result of sketching a line, arc or circle. For example, if you draw a line, two implicit end points are defined.

2 - Explicit - these points are created using the Sketch / Point menu option.

There are the following differences between the two types of points:

1 - Explicit points are graphically displayed as small circles, implicit points are not displayed.

2 - Explicit points must be explicitly selected to be Cut, implicit points are automatically Cut if all the other geometry depend ing upon them is Cut.

3 - Explicit points are created using Sketch / Point, implicit points are created as a result of sketching other entities.

Explicit points are very useful as locations to add point masses and applied loads and as points whose motion may be traced. Explicit points are also useful to mark the intersection between pairs of lines or circles.

To create a point:

1 - Pick the Sketch / Point toolbox icon.

2 - *Position the cursor at the location where you want to place a point.*

3 - *Click on the mouse button.*

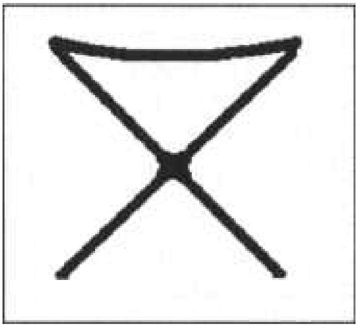
When you move the mouse over a line which already exists, that line will be highlighted and its name and length will appear in the Information box at the upper right of the Analytix window. If you click the mouse while a line is highlighted, the point will be constrained to lie on that line.

When you move the mouse over a circle or arc which already exists, that circle or arc will be highlighted and its name will appear in the information box along with its center and radius. If you create the point while the circle is highlighted, then the point will be constrained to lie on that circle.

When you move the cursor over the intersection between a pair of lines, or a pair of circles or a line and a circle, you will notice that both entities are highlighted. If you create a point while both entities are highlighted it will be constrained to lie on both.

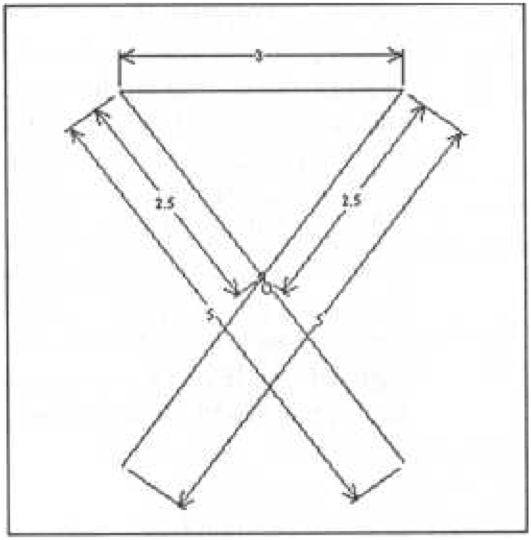
Example

To model a folding stool, we sketch three lines representing the two legs of the stool and its seat.



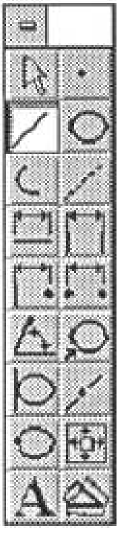
We can dimension the lengths of each leg and the length of the line representing the seat. To complete the dimensioning, we wish to dimension the position of the hinge on each of the legs. To do this we need to create a point at the intersection of the lines representing the legs.

This is done using Sketch / Point and ensuring both lines are highlighted at the place where the point is added.



Point to point dimensions can now be added between the line end points and the intersection point.

Sketch / Line



You sketch a line in Analytix as follows:

1 - Choose Sketch / Line from the toolbox icon.

2 - Position the cursor at the place you wish your line to start.

3 - Depress the left mouse button and, while keeping it depressed, drag the cursor to the place you wish the line to end.

4 - When you have reached the line end point, release the mouse button.

When you move the cursor over an already existing point in the drawing (either explicit or implicit) you will see the point's name and coordinates appear in the Information Box. If you release the mouse button while the cursor is positioned over the point, the line will adopt the indicated point as its endpoint.

When you move the cursor over a line which already exists, that line will be highlighted and its name and length will appear in the Information box at the upper right of the Analytix window. If you locate the line's endpoint while another line is highlighted, the endpoint will be constrained to lie on that line.

When you move the cursor over a circle or arc which already exists, that circle or arc will be highlighted and its name will appear in the information box along with its center and radius. If you release the mouse button while the circle is highlighted, then the endpoint will be constrained to lie on that circle.

When you move the cursor over the intersection between a pair of lines, or a pair of circles or a line and a circle, you will notice that both entities are highlighted. If you locate an endpoint while both entities are highlighted it will be constrained to lie on both.

Example

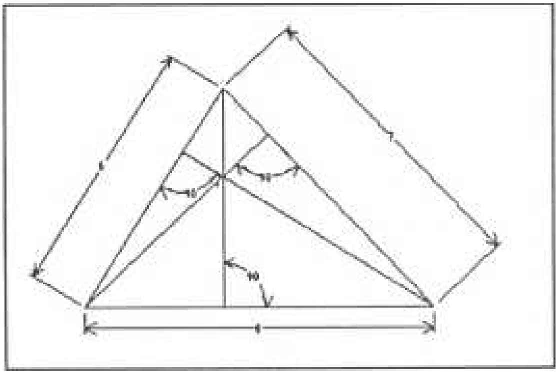
An elementary geometry theorem states that the altitudes of a triangle meet in a common point.

We cannot prove such a theorem with Analytix, but we can verify specific examples.

Draw a triangle. Dimension the lengths of its lines.

Now draw altitudes by sketching lines connecting vertices of the triangle to the opposite sides.

Ensure the altitude end points are constrained to lie on the triangle sides by making sure the sides are highlighted when the altitude endpoints are created.



Dimension the altitudes to be at 90 degrees to the respective triangle sides.

By inspection, note that the altitudes meet at a common point.

Of course, you can change the lengths of the sides of the triangles, and see that the altitudes still meet at a point. Do this three times, according to the story, and you have a physicist's proof of the theorem.

Sketch / Fillet

To sketch a fillet in Analytix simply -

Select the Sketch / Fillet menu option.

Click the mouse on the common endpoint of a pair of lines.

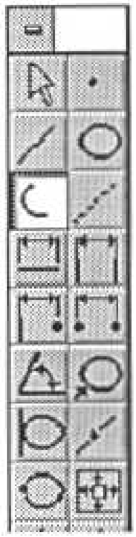
A fillet is really an arc tangent to each line whose endpoints match the endpoints of the lines.

The radius of the fillet sketched by Analytix is arbitrary, and chosen by the system to "look o.k."

The required fillet radius must be specified using the Dimension / Radius menu option.

Occasionally the addition of a fillet to an already consistently dimensioned figure will cause the figure to become over dimensioned. This is because adding a fillet adds three new dimensions to the system: a radius and two tangent dimensions. It is therefore best to draw fillets before completing the dimensioning of a figure.

Sketch / Arc



A circular arc is sketched in Analytix in the following way after selecting the arc toolbox icon:

1 - Position the cursor on the screen where one end of the arc is to be located.

2 - Press the left mouse button down and drag a chord to the position where the other end of the arc is to be located.

3 - When the chord is correctly positioned, release the mouse button.

4 - Move the cursor to a third point which should lie on the arc.

5 - Depress the left mouse button and drag the arc into the desired shape.

6 - When the desired arc shape is achieved, release the mouse button.

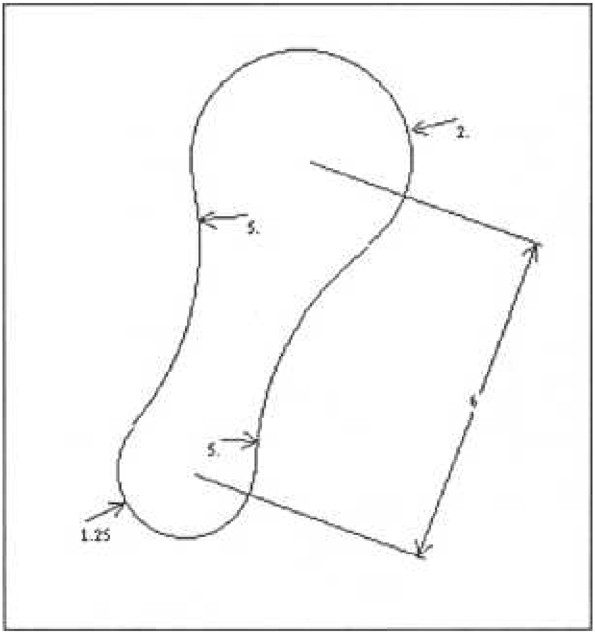
All arcs must have their radius dimensioned before a drawing may be consistently dimensioned.

Apart from fillets, arcs do not automatically become tangent to each other or to adjoining lines.

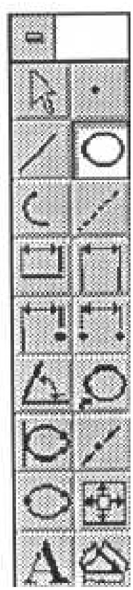
Tangencies must be explicitly set using the Tangent dimension.

Example

The picture shows a part comprising 4 joined arcs dimensioned by making the arcs tangent, by specifying the radius of each arc, and by specifying the distance between the centers of the two end arcs.



Sketch / Circle



Sketch a circle in Analytix by selecting the Circle toolbox icon:

1 - Position the cursor at the place where you want the center of your circle to be.

2 - Depress the left mouse button and drag the circumference of the circle.

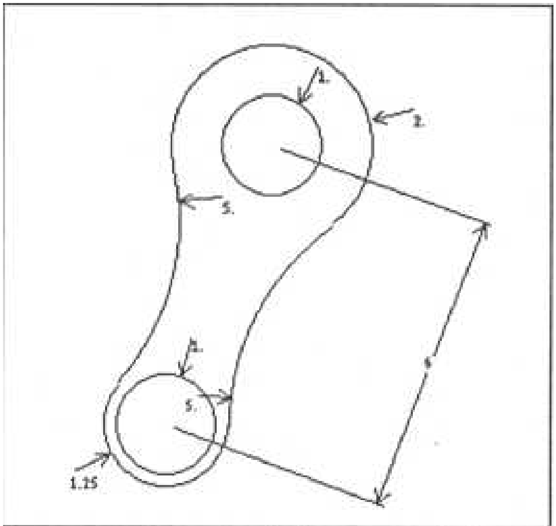
3 - When the radius of the circle looks right, release the mouse button.

When you move the cursor over an already existing point in the drawing (either explicit or implicit) you will see the point's name and coordinates appear in the Information Box. If you release the mouse button while the point is thus indicated, the circle will adopt the indicated point as its center.

When you move the mouse over a line which already exists, that line will be highlighted and its name and length will appear in the Information box at the top right of the Analytix window. If you release the mouse button while another line is highlighted, the center will be constrained to lie on that line.

When you move the mouse over a circle or arc which already exists, that circle or arc will be highlighted and its name will appear in the information box along with its center and radius. If you release the mouse button while the circle is highlighted, then the center will be constrained to lie on that circle.

When you move the cursor over the intersection between a pair of lines, or a pair of circles or a line and a circle, you will notice that both entities are highlighted. If you release the mouse button while both entities are highlighted the center of the circle



will be constrained to lie on both.

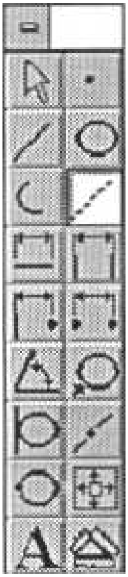
Example

We take the part pictured above as the Arc drawing example and draw two holes concentric with the arcs at either end of the arm.

We do this by drawing two circles, but ensuring that the mouse was over the center of the arcs when we position the center of our circles. To ensure the mouse is correctly located, we verify that the point information is displayed in the Information box.

We now need only dimension the radii of the new circles to have a fully dimensioned part.

Sketch / Construction Line



In Analytix, a construction line differs from an ordinary line in two ways.

1 - It is usually drawn in a different line style.

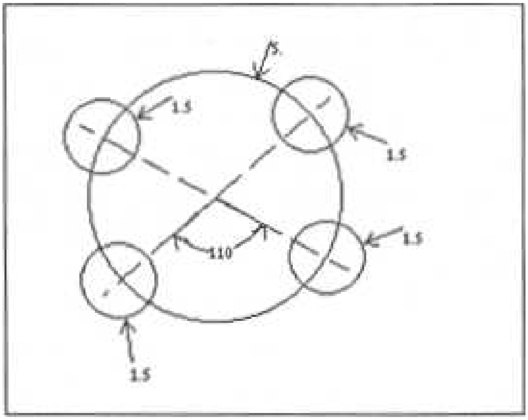
2 - It has no specific end points, rather it is automatically sized to overlap the current figure a little.

Drawing a construction line is identical to drawing any other line, except the endpoints will not become automatically constrained to lie on any point, line, arc or circle. (Other points, however, may be constrained to lie on the construction line.)

End points of construction lines may not be dimensioned. However the construction line itself may be dimensioned (using Angle, Distance Line to Point, or Parallel Distance).

Example

We construct a hole pattern.



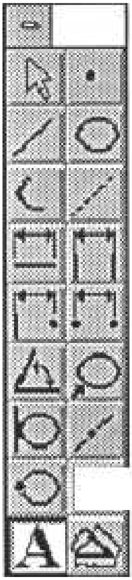
1 - Draw two construction lines as shown.

2 - Draw a circle whose center lies at the intersection of the construction lines.

3 - Draw 4 smaller circles at the intersection of the construction lines and the larger circle.

4 - Dimension the figure by specifying the radii of all the circles and the angle between the construction lines.

Sketch / Annotation



Annotation in Analytix can be positioned absolutely on the drawing, or relative to a point of the drawing, or a line of the drawing.

Absolute annotation stays at the specified location, even when the drawing moves.

Relative annotation, on the other hand maintains its position relative to the geometry when the geometry moves.

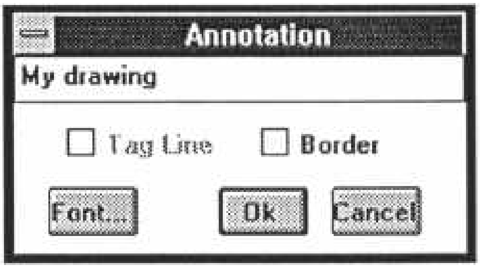
Relative annotation is good for labeling geometry or for displaying information about some piece of the geometry. Absolute annotation is good for providing information about the drawing as a whole.

Annotation may include text; it may also include expressions in between @ symbols. These expressions are sent to the calculator and evaluated, then displayed. This lets you display analysis results as annotation on the drawing.

Absolute Annotation

To create absolute annotation, select Sketch 1 Annotate, or click on the Annotation Icon of the Toolbox.

An Annotation dialog box now appears.



Type the text of the annotation into this box. You may select the corresponding button to place a border round the annotation, or to change the font of the annotation.

Once an annotation has been created, it may be selected and moved by clicking on it and dragging it. An annotation may be edited by selecting it then using the Attributes / Info menu option, or by double clicking on the annotation.

Relative Annotation

Annotation may be specified relative to a point, a line or a point and line.

If the annotation is relative to a point and line, the location of the top left comer of the annotation is measured in a coordinate system whose origin is at the point and whose x axis is aligned with the line and scaled to the length of the line.

As the line rotates or the point moves or the line grows or shrinks, the top left corner of the annotation moves accordingly.

Annotation relative to a line is equivalent to annotation relative to one end point and the line itself.

Annotation relative to a point is equivalent to annotation relative to the point and a unit line in the x direction.

To specify relative annotation:

Select the point, line or point and line to which you want to attach the annotation.

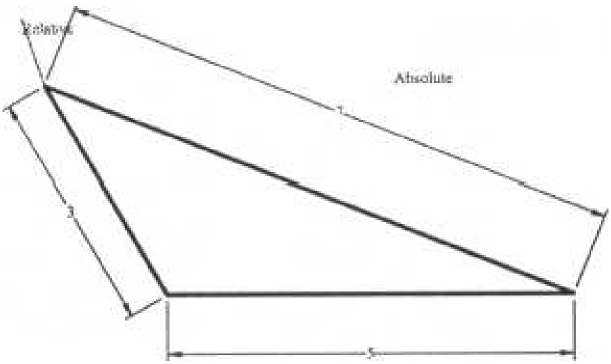
Select Sketch / Annotate or the Annotation Icon in the Toolbox.

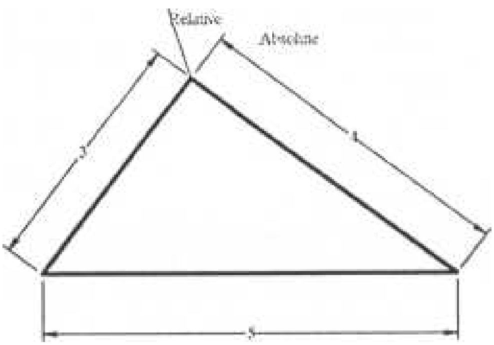
Click the location on the screen where you want the annotation.

The Annotation dialog box appears.

Type the text of the annotation in this box. You may also specify a border and a tag line.

The tag line joins the annotation to its referenced geometry.





Expressions in Annotations

Any expression which is typed between @ symbols in the text of the annotation will be evaluated by the calculator and the resulting number displayed on the screen.

For example if a is a variable in the calculator with a value of 2.5, the following annotation:

will be displayed as follows:

a=2.5

If DIMENSION23 has a reaction force of -1.275, the following annotation:

Reaction force is @react(DIMENSION23)@

will be displayed:

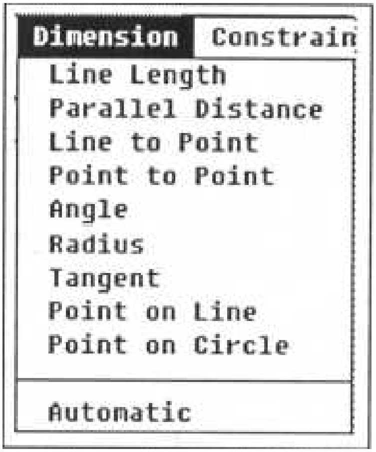
Reaction force is -1.275

If POINT3 has coordinates (1.7,3.5), the following annotation:

will be displayed:

(1.7,3.5)

Dimension



When you have entered a sketch into Analytix, the way to convert that sketch to a scale drawing is to add a consistent set of dimensions to the sketch. You do this using the Dimension menu.

A good analogy is with the way drawings were made before CAD. The engineer would communicate his design to the drafting department using a not-to-scale dimensioned sketch. The sketch indicates the overall shape of the object, but the dimensions give the draftsman the precise distances and angles to use. However, if there were either too many or too few dimensions to correctly define the part, it would come back from the draftsman for clarification.

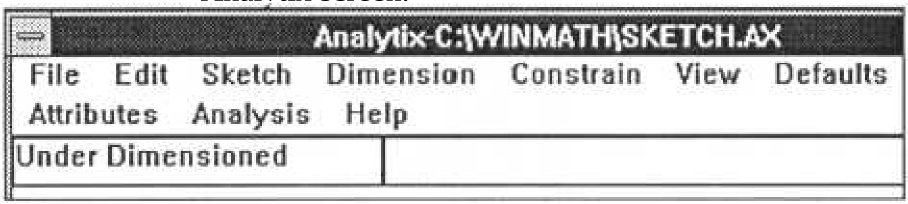
Analytix works in a similar way

1 - Sketch your part without worrying if your sketch is exactly right, but ensuring

the sketch does in fact resemble the intended part.

2 - Add dimensions to indicate the exact geometry.

Analytix will tell you whether your part is under dimensioned, consistently dimensioned, or has a redundant dimension. This appears in the Dimension Status Box in the upper left corner of the Analytix screen.



When the sketch is consistently dimensioned, Analytix will automatically produce a scale drawing from the dimensioned sketch representation.

While the sketch is under dimensioned, Analytix makes no attempt to alter the drawing to match the constraints already entered. It simply waits until it has a consistently dimensioned drawing. This is so that when it does come to create a scale drawing it has your original, unaltered sketch to work with.

If, however, you have entered all the dimensions you care about, you can ask Analytix to finish off the dimensioning for you using the Dimension / Automatic menu option. Analytix will create a consistent dimensioning scheme for your drawing which includes the dimensions which you have already specified.

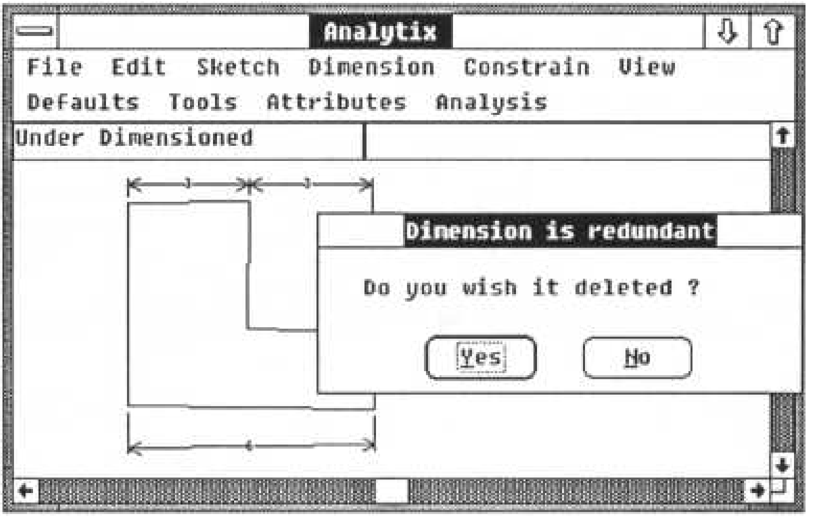
Redundant Dimensions

There are two situations where Analytix will declare a dimension to be redundant:

I - When the distance or angle to be specified by the dimension is already determined by some other combination of dimensions.

2 - When it is not possible to construct the drawing from the dimensions using ruler, protractor, compasses and parallel rules.

In either case, Analytix will give you the option of removing the redundant dimension. If you do not do this, you will need to remove some other dimension before you can proceed to create a consistently dimensioned drawing.



1. Dimension value already known

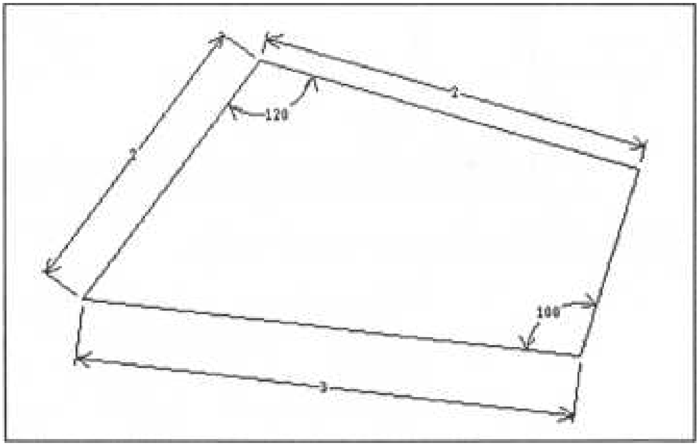
In the picture, all three dimensions are parallel distances. They obviously comprise a redundant set, as the fact that the larger distance is 6 can be derived from the fact that each of the smaller distances is 3.

If you click on the Yes button, Analytix will delete the most recently added dimension. If you click on the No button, your drawing will remain redundantly dimensioned. You could delete any of the lengths given to return to a non-redundant situation.

2. Figure not constructible

Analytix uses Constructive Variational Geometry to convert a dimensioned drawing into a scale drawing. Constructive Variational Geometry is the computer equivalent of a human making a drawing with ruler, compasses, protractor and parallel rules. If it is not possible to draw the figure using only these tools, Analytix will not be able to construct a drawing either.

An example of a figure which is not constructible (although well defined geometrically) is a quadrilateral defined by three side lengths and two opposite angles. There is no way to construct this figure from the dimensions using the prescribed tools - go ahead, try it. You would have to "trig it out" : use the given lengths and angles to calculate, perhaps, one of the other angles.



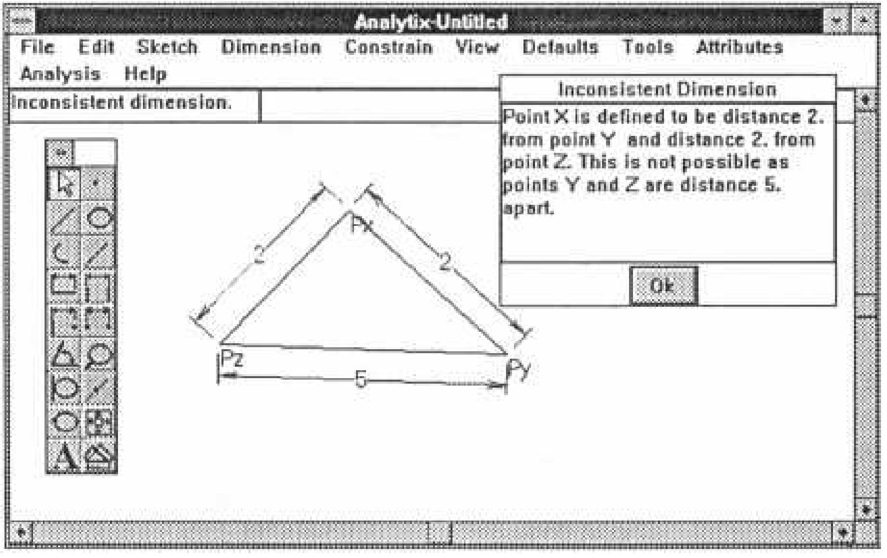
There are two ways to enter the above part into Analytix:

1 - Trig it out and calculate the value of some other dimension and enter that.

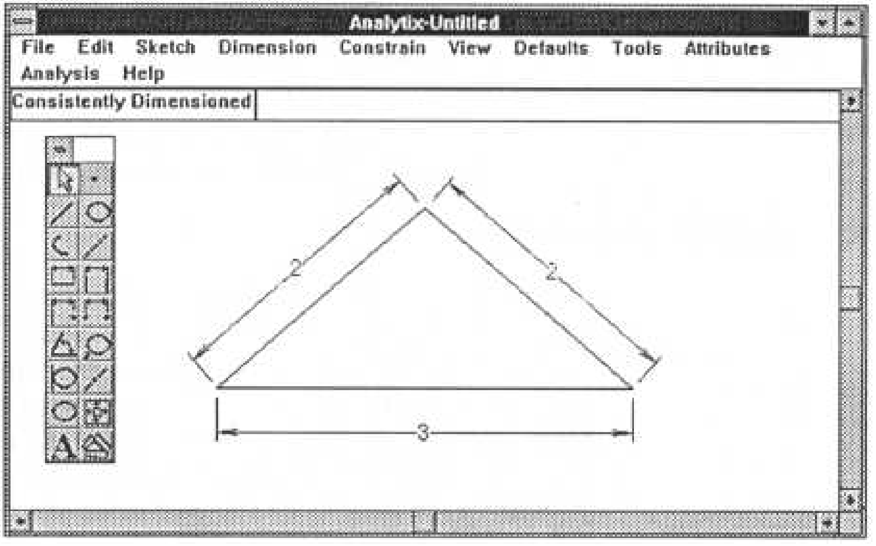
2 - Use the iteration tool. Delete, say, the 100 degree dimension and add a dimension at the angle between the line of length 2 and the line of length 3. Now iterate on the value of this dimension so that the required angle comes out to be 100 degrees. (See the documentation under the Tools / Univariate Iteration menu option.)

Inconsistent Dimensions

A drawing may have dimensions which are mutually independent - no one dimension is known as a function of the others - but it still may not be possible to draw the figure. For example a basic geometrical condition such as the triangle inequality may be broken. [The triangle inequality states that the sum of the lengths of any two sides of a triangle must be greater than the third side].



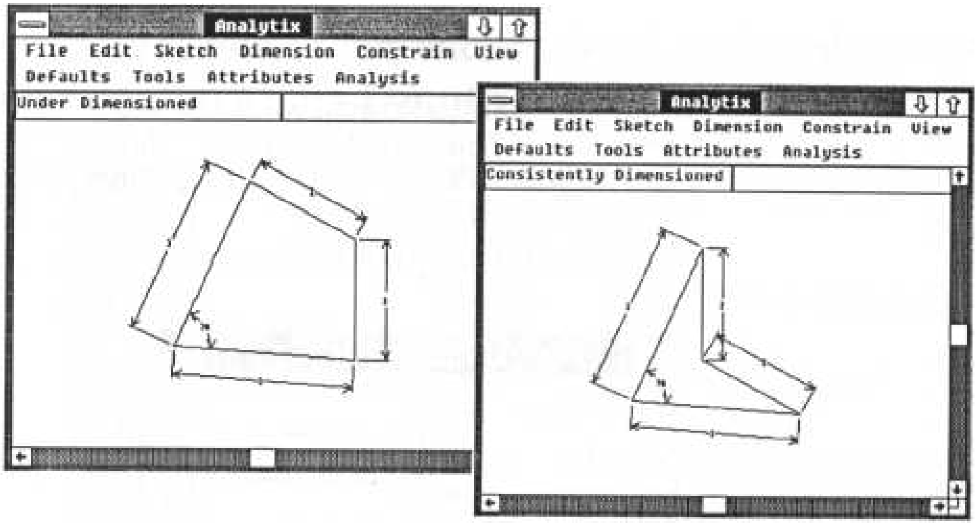
If there is such an inconsistent dimension, then Analytix is unable to create a scale drawing from the sketch. You should go back and change the values of one or more dimensions to make the figure drawable.



Multiple solutions

Some dimensioned sketches have more than one dimensioned drawing which satisfies the geometry. For example, there are two different representations of a quadrilateral with the lengths of each side and a single angle dimensioned. One is convex, the other is not.

Analytix decides which solution to use based on the sketch. In all cases the criteria used by the system are simple geometrical ones. For example in the quadrilateral case, if the quadrilateral as drawn is convex, then the solution used by Analytix is convex. Otherwise, the solution used by Analytix is concave.



If you find that Analytix has used the wrong solution, you need to reshape your sketch to look more like the solution you had in mind. You can do this by dragging single points using the Select-and-Drag option or by Moving or Rotating whole segments of the drawing using the View / Move and View / Rotate menu options.

Expressions as Dimension Values

Instead of entering a number as the value of a dimension, you can enter any legal Analytix expression. You could for example set the value of a dimension to be:

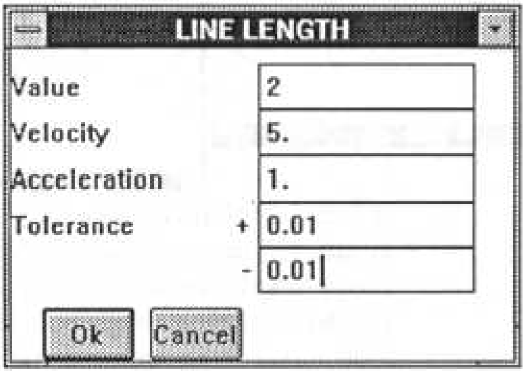
- sqrt(3.5^2 + 4.75^2)
- sin(t)
- 2*(a+b)
- c=3.5

Assuming, in the above that variables a, b and t have been defined either in some other dimension in the drawing or in the Calculator. (See Tools / Calculator.)

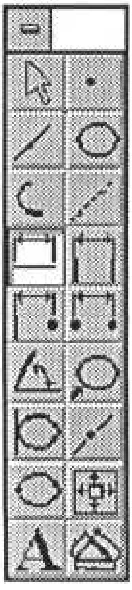
Setting Velocities, Accelerations and Tolerances

To use the sophisticated Statics, Kinematics, Dynamics and Tolerance Analysis capabilities of Analytix, you must give velocities, accelerations and Tolerances to dimensions.

You can do this using the Attributes / Info menu option.



Dimension / Line Length



The Line Length dimension lets you specify the length of a line.

To enter a line length dimension:

1 - Choose the Dimension/Line Length toolbar icon.

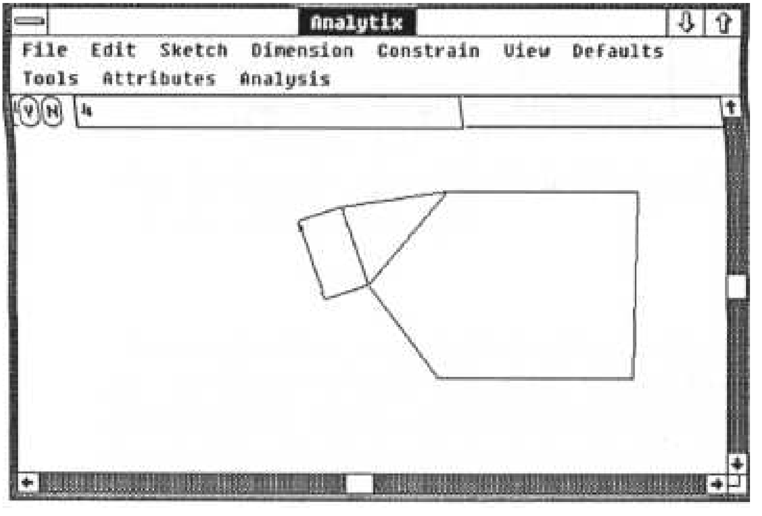
2 - Position the cursor over the line to be dimensioned.

3 - Press down on the left mouse button and drag the cursor away from the line. You will notice a dimension symbol following the cursor.

4 - When the dimension symbol is in an appropriate place, release the mouse button.

5 - Type in the dimension value in the Dimension Entry Box.

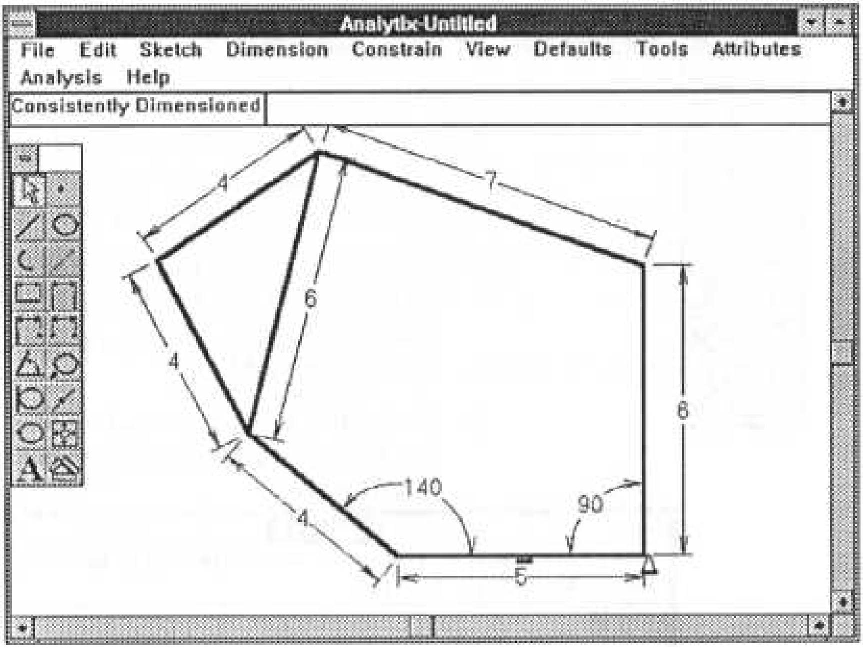
6 - Complete the dimension value entry by clicking on the Y button, or by pressing the [ENTER] key.



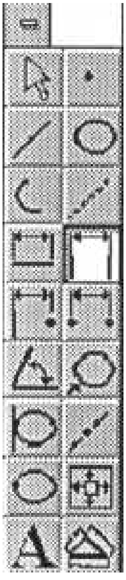
Note that you cannot give a length dimension to a construction line, as the construction line has by definition no specific length.

Example

The example shown is a linkage dimensioned by specifying the lengths of all the lines in the mechanism, and two angles.



Dimension / Parallel Distance



The parallel distance dimension specifies that two lines are parallel and it specifies their distance apart.

This is the dimension which will be heavily used in defining parts which are comprised mainly of horizontal and vertical line segments.

To make a Parallel Distance dimension:

1 - Choose the Dimension/Parallel Distance tool box icon.

2 - Click on the first line to be dimensioned.

3 - Position the cursor over the second line to be dimensioned.

4 - Press down on the left mouse button and drag the cursor away from the line. You will notice a dimension symbol following the cursor.

5 - When the dimension symbol is in an appropriate place, release the mouse button.

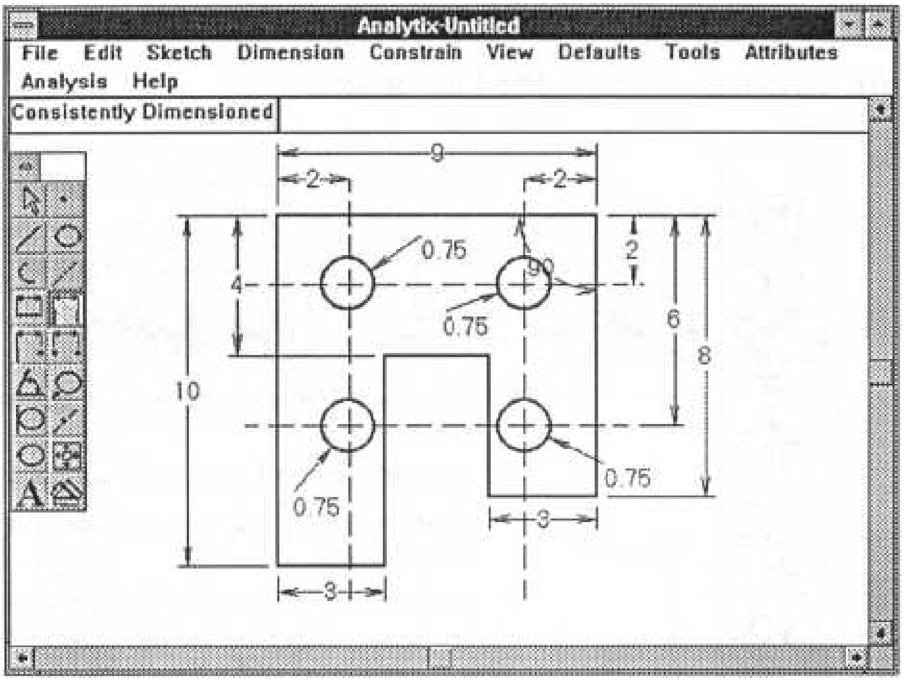
6 - Type in the dimension value in the Dimension Entry Box.

7 - Complete the dimension value entry by clicking on the Y button, or by pressing the [ENTER] key.

You can dimension construction lines using the parallel distance dimension.

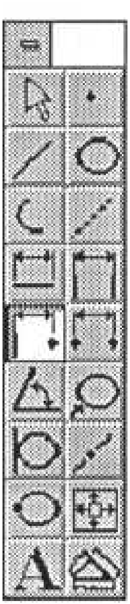
Example

The example shows a part dimensioned using only parallel distance dimensions, along with one angle and four radii. (Along with the fact that the circles were constructed to lie on the intersections of the construction lines.)



Analytix-Untitled
File Edit Sketch Dimension Constrain View Defaults Tools Attributes
Analysis Help
Consistently Dimensioned
9
2
2
0.75
90
2
4
0.75
6
8
10
0.75
0.75
3
3

Dimension / Line to Point



This dimension lets you specify the perpendicular distance between a line and a point.

To make a Distance Line to Point dimension:

1 - Choose the Dimension / Line to Point tool box icon.

2 - Click either on the line or point to be dimensioned (or on the circumference of a circle or arc if its centerpoint is to be dimensioned)

3 - Position the cursor over the second entity to be dimensioned.

4 - Press down on the left mouse button and drag the cursor. You will notice a dimension symbol following the cursor.

5 - When the dimension symbol is in an appropriate place, release the mouse button.

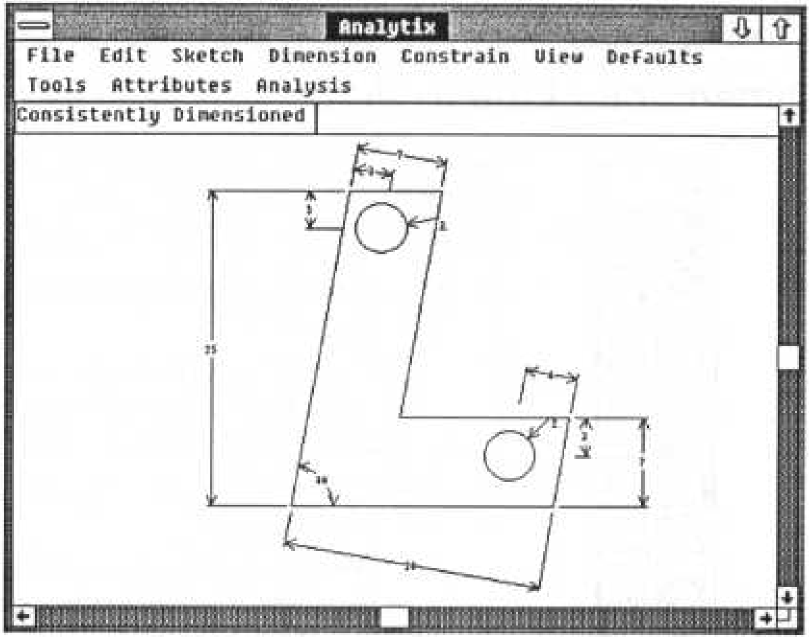
6 - Type in the dimension value in the Dimension Entry Box.

7 - Complete the dimension value entry by clicking on the Y button, or by pressing the [ENTER] key.

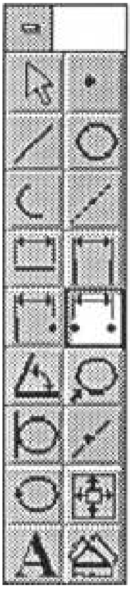
Construction lines may be dimensioned using the Line to Point dimension, however the end points of construction lines may not be dimensioned since they have no end points.

Example

In the example, the centers of the circles are positioned by Distance Line to Point dimensions specifying their distance from the edges of the part.


Analytix
File Edit Sketch Dimension Constrain View Defaults
Tools Attributes Analysis
Consistently Dimensioned

Dimension / Point to Point



The Point to Point dimension lets you specify the distance between two points.

If the points happen to be the endpoints of a line, this dimension is the same as the Line Length dimension applied to that line.

To make a Distance Point to Point dimension:

1 - Choose the Dimension / Point to Point tool box icon.

2 - Click one of the points to be dimensioned (or on the circumference of a circle or arc if its center point is to be dimensioned)

3 - Position the cursor over the second point to be dimensioned (or on the circumference of a circle or arc if its center point is to be dimensioned).

4 - Press down on the left mouse button and drag the cursor. You will notice a dimension symbol following the cursor.

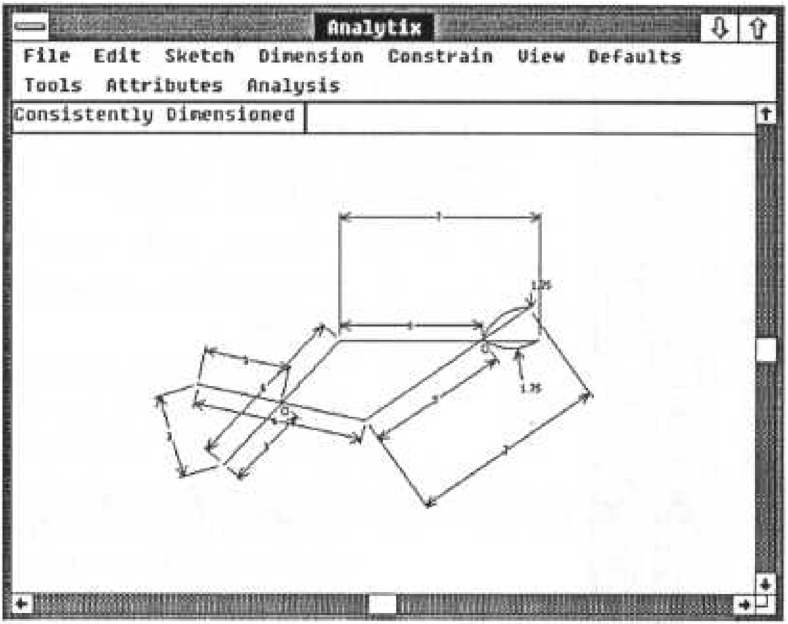
5 - When the dimension symbol is in an appropriate place, release the mouse button.

6 - Type in the dimension value in the Dimension Entry Box.

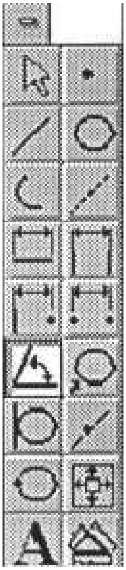
7 - Complete the dimension value entry by clicking on the Y button, or by pressing the [ENTER] key.

Example

In this model of a pair of snips, points were sketched at the two line intersections. These points were dimensioned using Distance / Point to Point dimensions. The separation of the two handles of the snips is also specified using the Distance / Point to Point dimension type.



Dimension / Angle



This lets you specify the angle between two lines. The lines do not need to share a common endpoint..

To make an Angle dimension:

1 - Choose the Dimension /Angle tool box icon.

2 - Click on the first line to be dimensioned.

3 - Click on the second line to be dimensioned.

4 - The angle dimension symbol will appear in either the acute or obtuse angle between the lines.

5 - Type in the angle value in the Dimension Entry Box.

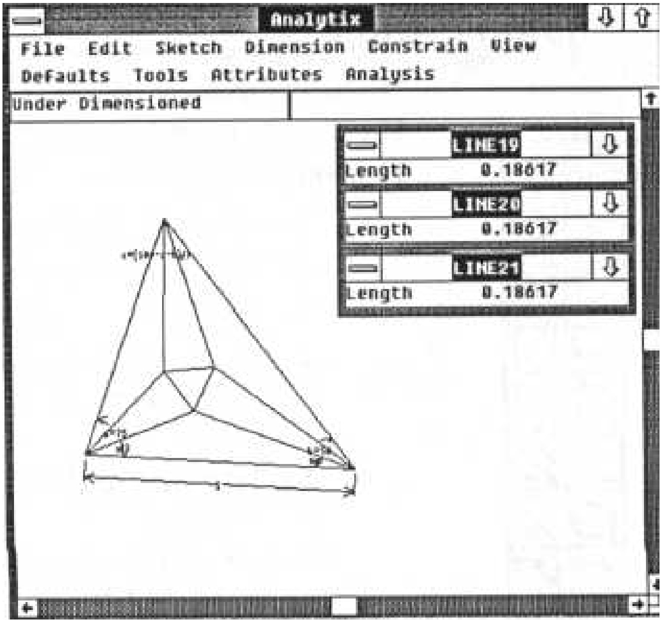
6 - Complete the dimension value entry by clicking on the Y button, or by pressing the [ENTER] key.

Construction lines can be given angular dimensions.

Example

In this example we illustrate a geometrical theorem due to the physicist Roger Penrose. It says that the triangle formed by the trisectors of the angles of any triangle is equilateral.

It is interesting that such an elegant theorem in elementary geometry should have remained undiscovered until this century. One reason for this is that the Greeks thought of geometry in terms of constructions using straight edge and compass. The problem of trisecting an angle using these tools was a classic unsolved problem, which was finally shown to be impossible in the 19th century. Hence mathematicians had not gone beyond trying to construct angle trisections to see what properties such trisections would have.

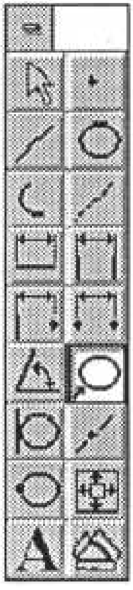


Our triangle is dimensioned by the length of its base and the angles at either end of the base. These angles are entered as a=75 and b=50. Angles between the triangle sides and the trisectors are entered as a/3 , b/3, and (180-a-b)/3. Two out of three angles in each set need to be dimensioned: the third one comes for free.

With these dimensions, our figure is consistently dimensioned. We can measure the length of the sides of the small triangle in the center of the figure, using the Attributes / Info menu option (or by double clicking the right mouse button).

As stated by the theorem, all three sides are equal. We can alter the shape of the original triangle by changing a and b and verify that the theorem still holds.

Dimension / Radius



This lets you specify the radius of an arc or circle.

In Analytix, all arcs circles and fillets must be given an explicit radius.

To specify the radius of an arc or circle:

I - Select the Radius icon from the tool box.

2 - Position the cursor over the arc or circle.

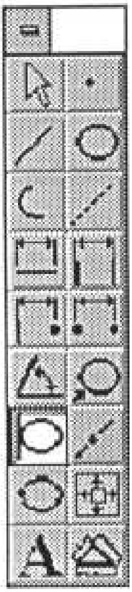
3 - Press the left mouse button and drag the dimension symbol to an appropriate position on the curve.

4 - When the radius symbol is correctly placed, release the mouse button.

5 - Type the radius value in the dimension entry box.

6 - Click on the Y button or press [ENTER].

Dimension / Tangent



This lets you specify that a circle or arc is tangent to a line or to another circle or arc.

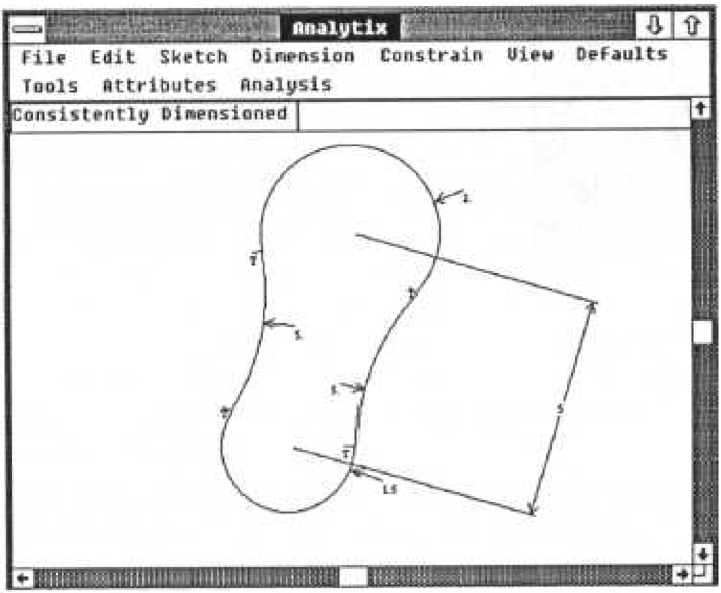
To specify that two entities are tangent:

1 - Select the Tangent icon from the tool box.

2 - Click on the first entity (line, circle or arc).

3 - Click on the second entity.

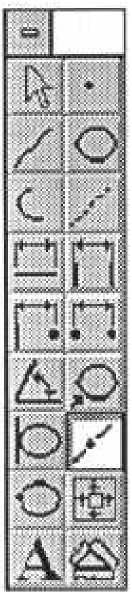
Unless the tangency creates a redundant dimension, the entities are now tangent.



Example

The four arcs in the example are tangent. The whole figure is dimensioned by the radius of the arcs and the distance between the centers of the end arcs.

Dimension / Point on Line



This lets you specify that a given point lies on a given line.

To specify a Point on Line dimension:

1 - Select the Dimension / Point on Line icon from the tool box.

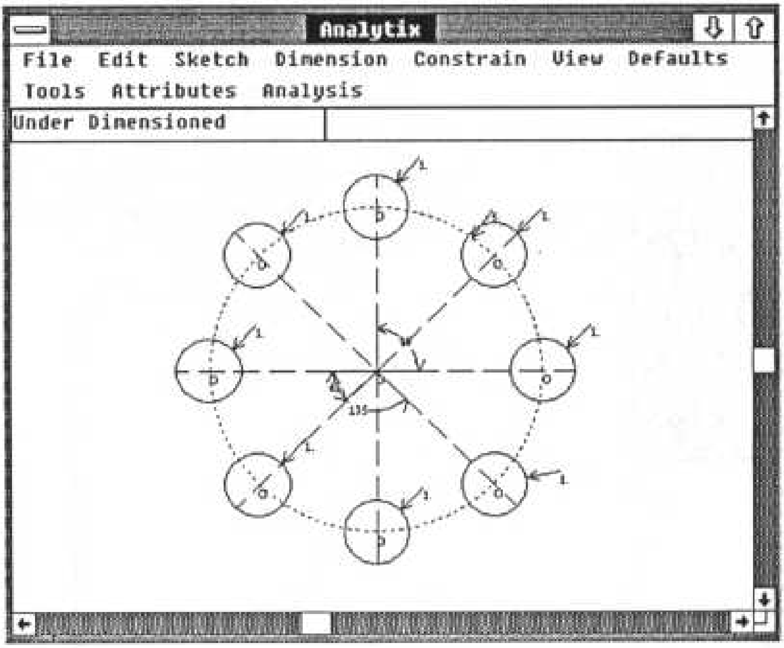
2 - Click on the first point or line.

3 - Click on the second line or point.

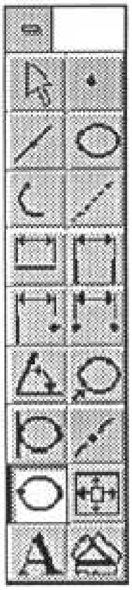
Construction lines may be dimensioned using the Point on Line dimension, but their end points may not as they have no end points.

Example

In the example, the circle center is located at the intersection of two of the construction lines. The Point on Line dimension is used to constrain the other two construction lines to pass through the center of the circle.



Dimension / Point on Circle



This lets you specify that a given point lies on a given circle.

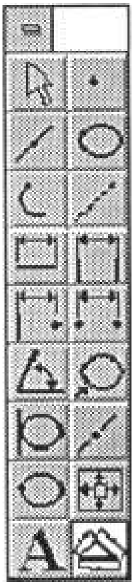
To specify a Point on Circle dimension:

1 - Select the Dimension / Point on Circle icon from the tool box.

2 - Click on the first point or circle.

3 - Click on the second circle or point.

Dimension / Automatic



This menu option tells Analytix to complete the dimensioning of the drawing itself.

This is a very powerful feature and is of particular use in these circumstances:

- When you do not know what else to dimension.
- When you have read in from file the correct geometry for a part and do not care how it is dimensioned.
- When you have entered all the dimensions you are currently concerned with and wish Analytix to fill in the rest.

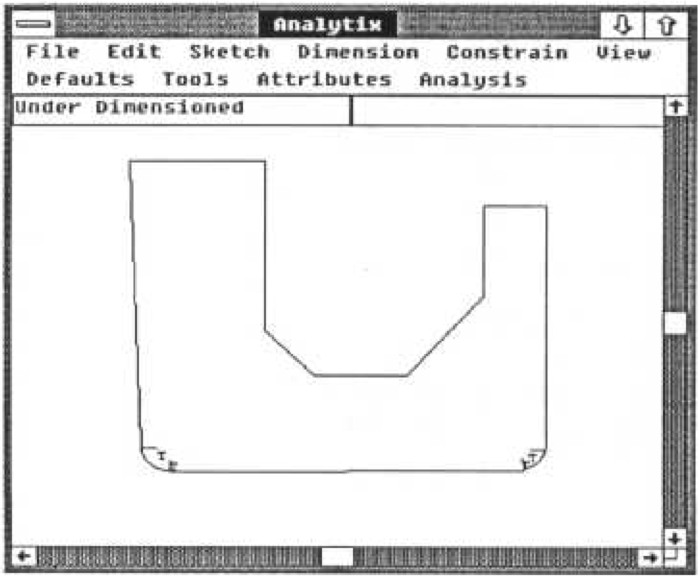
There are many different ways to consistently dimension a given drawing. Analytix has a suite of heuristic algorithms which try to find "reasonable" dimensions to fill in. However there is no guarantee that the dimensions it uses are the ones which you want.

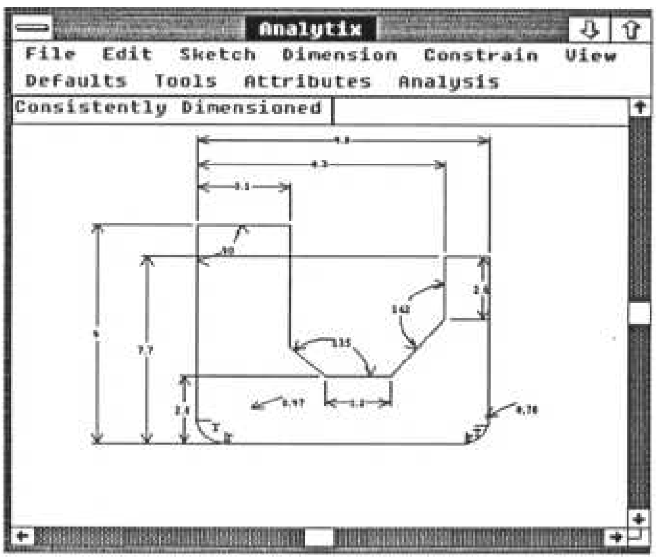
Analytix fills in a value for the dimensions which it creates by measuring from the sketch and scaling to make them match any dimensions which have already been entered. The value thus achieved is then rounded.

Example

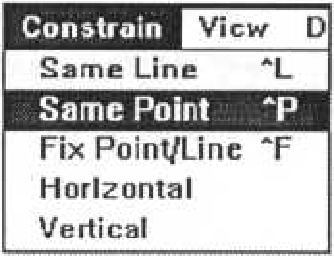
This part was dimensioned completely by Analytix.

First a sketch was entered, then Dimension / Automatic was invoked.





Constrain



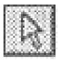
The Constrain menu has options which let you constrain line segments to be collinear, circles and arcs to be concentric, and to set which point and line of the figure is to stay fixed.

Constrain / Same Line

Sets the selected line segments to be collinear.

This option is grayed out unless the current set of selected entities comprises two or more lines.

To constrain a number of line segments to be collinear:



1 - *Make sure you are in Selection mode, either by picking the Edit / Select icon from the tool box or by clicking the right mouse button.*

2 - *Click on the first line to be constrained.*

3 - *Holding down the shift key, click on the second line to be constrained.*

4 - *Repeat step 3 for the rest of the lines to be constrained.*

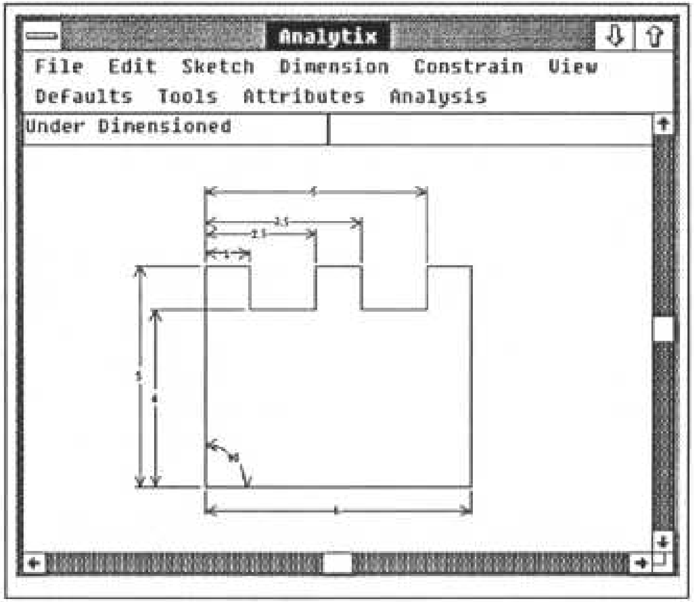
5 - *When the correct set of lines is selected, pick Constrain / Same Line.*

If the addition of the Same Line constraint causes the drawing to be over dimensioned, then you will see the Redundant Dimension message. Otherwise the lines will be constrained to be collinear.

To remove a collinear constraint, you must delete the line, then redraw it.

Example

In the example, the three lines at the top of the tabs are made collinear; the two lines between the tabs are made collinear. The part is then dimensioned using Parallel Distance dimensions and a single angle.

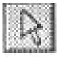


Constrain / Same Point

Sets the currently selected points and/or circle centers to be identical.

This option is grayed out unless the current set of selected entities comprises two or more points, arcs or circles.

To constrain a number of points to be the same:



1 - Make sure you are in Select mode, either by picking the Edit / Select icon from the tool box or by clicking the right mouse button.

2 - Click on the first point to be constrained (Or click on the circumference of a circle or arc to constrain its center).

3 - Holding down the shift key, click on the second point to be constrained (Or click on the circumference of a circle or arc to constrain its center).

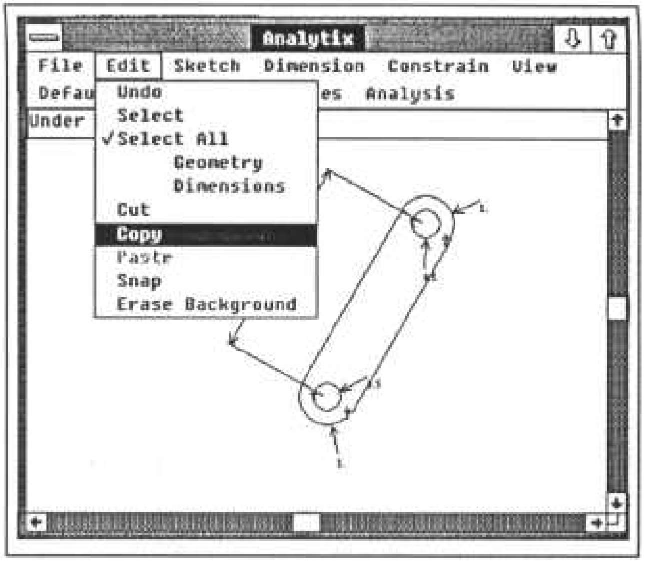
4 - Repeat step 3 for the rest of the points to be constrained.

5 - When the correct set of points is selected, pick Constrain / Same Point.

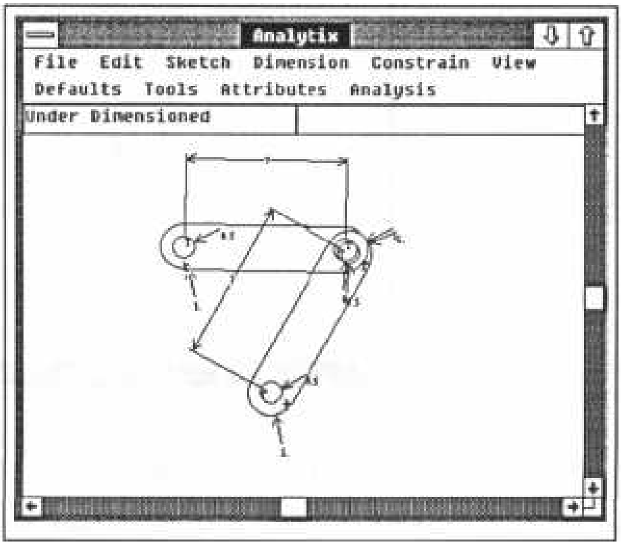
If the addition of the Same Point constraint causes the drawing to be over dimensioned, then you will see the Redundant Dimension message. Otherwise the points will be constrained to be the same.

Example

In this example, we construct a model of a four bar linkage, where each link is a bar with semi-circular ends and circular pin holes.



First construct a prototype bar by sketching two parallel lines and arcs to join the ends of the bar. Now sketch circles making sure the circle center is the arc center.



Add tangency dimensions between the arcs and the lines, radii for all arcs and circles, and the distance between the circle centers.

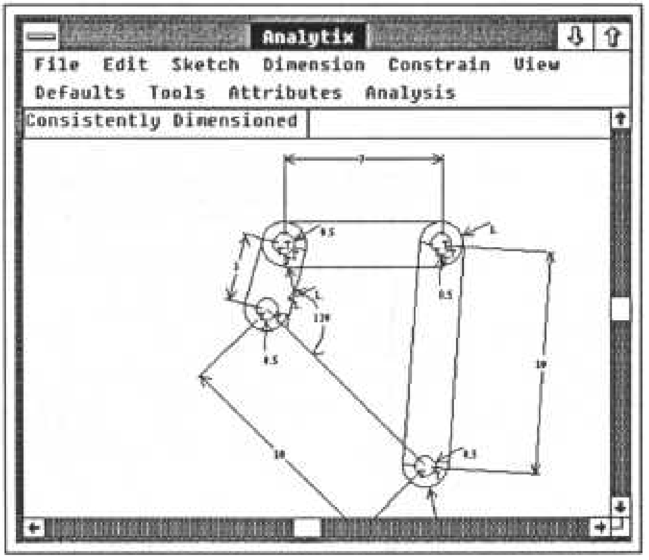
Now Copy this entire part to the Clipboard, then Paste a second copy onto the screen.

Use View / Move and View / Rotate to position the second copy.

Paste another copy to make the third link, and draw a line joining the first and third link to form the base.

We are now ready to complete the dimensioning of the linkage.

We do this by dimensioning the length of the ground line, by changing the lengths of the links as appropriate, and by "glueing" the centers of the circles at the end of links together using the Same Point constraint.



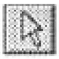
If we now fix the baseline and add the angle between the driving crank and the baseline, we have a working linkage.

Constrain / Fix Point/Line

This option lets you specify a point and a line of the drawing to be fixed. The fixed point must lie on the fixed line.

This menu option is grayed out unless either a point, a line or a point and a line are currently selected.

To specify the fixed point and line:



1 - Make sure you are in Select mode, either by picking the Edit / Select icon from the tool box or by clicking the right mouse button.

2 - Select the point (or line) to be fixed.

3 - (Optionally) select the line (or point) to be fixed.

4 - Choose the Constrain / Fix Point/Line option.

A pin icon will appear at the fixed point and a roller icon will appear in the middle of the fixed line. You may have at most one fixed point and one fixed line in your drawing.

When is it important to fix a point and line

When velocities and accelerations are specified in Analytix, they are always relative. In order to convert these velocities and accelerations to absolutes, you need to tell Analytix what is fixed and what is moving.

Hence, for dynamic and kinematic analysis, it is necessary to specify a fixed point and line.

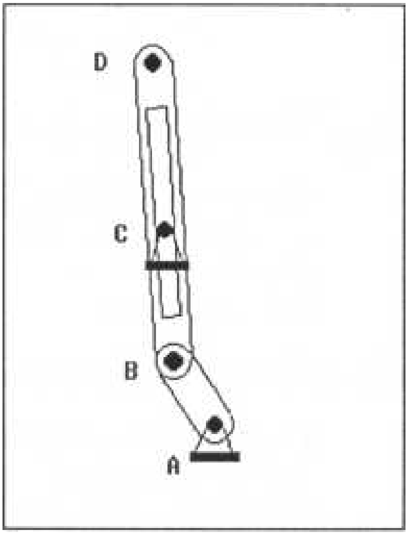
Many static problems assume some points of the figure are anchored, for these problems it is necessary to fix a point and line.

If you use the tolerance zone analysis, you should specify a fixed point and line. These will then be assumed to be in true position and tolerance zones of other points will be displayed.

If you want to measure velocities and accelerations with respect to some moving (even accelerating) frame of reference, all you need to do is specify a point and line in that frame as being fixed.

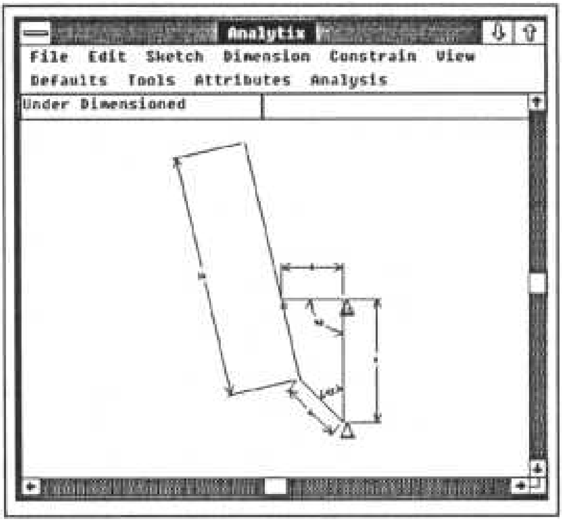
If there are several fixed points in your drawing, you need only specify one as fixed, and fix a line joining two of the points. Additional fixed points are constrained by dimensions.

Example

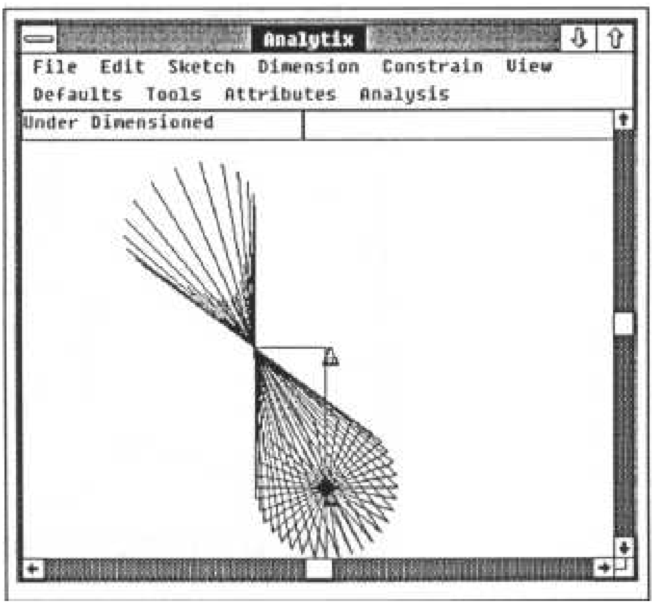


In this worked example, we wish to determine the envelope of the slider mechanism shown above. We also wish to determine the velocity of point C along bar BD when the crank AB is at 120 degrees to the horizontal.

AB is length 4, BD is length 16, C is distance 8 above A and distance 2 to the left. Crank AB rotates counter-clockwise at 24 rad/s.



To draw the envelope of the mechanism, we must ensure that points A and C stay fixed while the rest



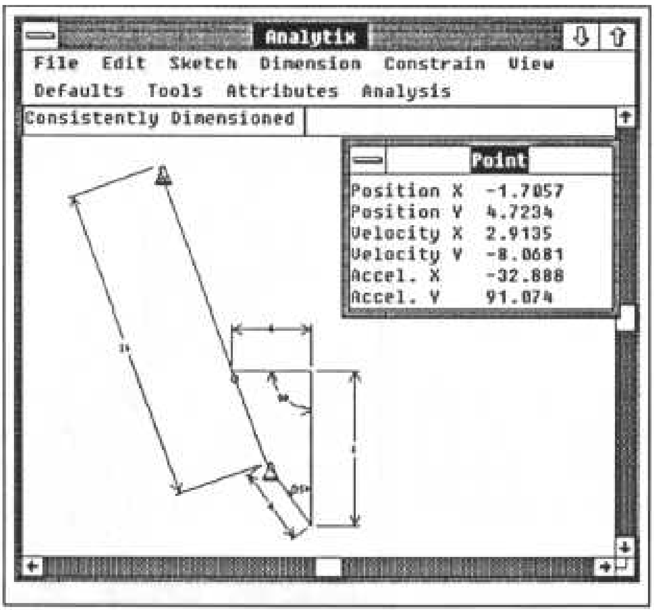
of the mechanism moves. In Analytix, we cannot set two points to be fixed - as then Analytix would not know what to do if the distance between the points was caused to alter by the changing of a dimension.

We therefore set point A to be fixed and a vertical line through A. As none of the dimensions which define the position of C with respect to A change over the course of the motion, C will also stay fixed.

To complete the second part of the problem, we need to calculate the velocity of point C relative to the moving bar BD.

To do this we simply make the bar BD stationary by fixing point B and line BD. (When fixing a new point and/or line, any previously fixed point or line is automatically released.)

Assuming we have already entered the velocity of the angle at A, we use Attributes / Info to measure the velocity of point C. As line BD is stationary, this is the velocity of C with respect to BD



Constrain / Horizontal

If you fix a line in using Constrain / Fix Point/Line, its orientation will be exactly the orientation in the sketch.

You can use the Constrain / Horizontal option to specify that the selected line is fixed and horizontal.

First, from select mode, click the line to be fixed; then choose Constrain / Horizontal from the menu.

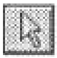
In Analytix you can have at most one fixed line. Hence if you already have a fixed line before selecting Constrain / Horizontal, then the existing fixed line will be freed up.

In Analytix, your fixed point must lie on your fixed line. Hence if you have a fixed point specified which does not lie on the selected line when you choose Constrain / Horizontal, then it will be freed up.

Constrain / Vertical

If you fix a line in using Constrain / Fix Point/Line, its orientation will be exactly the orientation in the sketch.

You can use the Constrain / Vertical option to specify that the selected line is fixed and vertical.

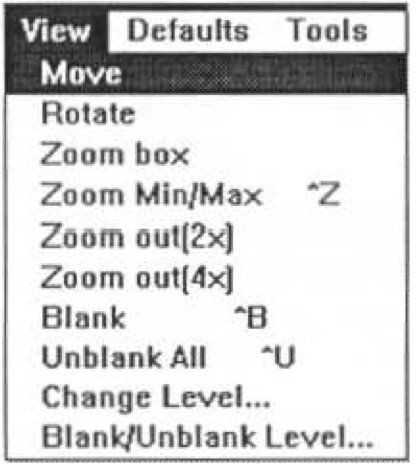


First, from select mode, click on the line to be fixed; then choose Constrain / Vertical from the menu.

In Analytix you can have at most one fixed line. Hence if you already have a fixed line before selecting Constrain / Vertical, then the existing fixed line will be freed up.

In Analytix, your fixed point must lie on your fixed line. Hence if you have a fixed point specified which does not lie on the selected line when you choose Constrain / Vertical, then it will be freed up.

View

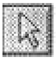


The View menu contains options to let you Move and Rotate selected objects, Zoom in or out on the drawing and Blank selected objects. It also has options to perform level management.

View / Move

Lets you move the currently selected portion of the sketch.

To move a portion of your drawing:



1 - Select the geometrical entities to be moved. (Dimensions on their own are not moved, but follow the geometry.) To select multiple entities use the shift key.

2 - Choose Move from the View menu.

3 - Position the cursor over the entities to be moved.

4 - Press down on the left mouse button and drag the entities to their new position.

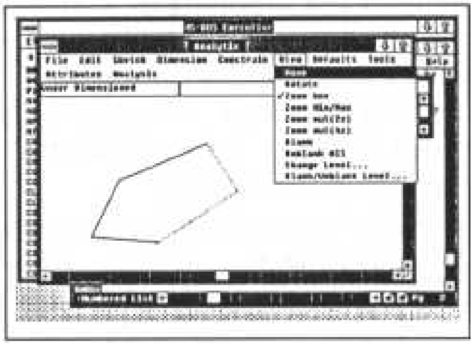
5 - When the new position is reached, release the mouse button.

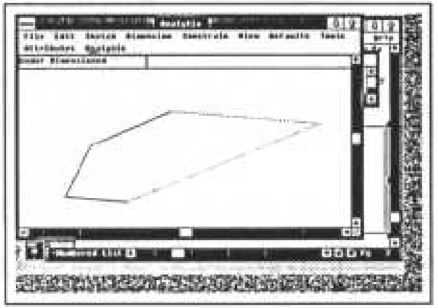
What is moved

If your set of selected entities shares points with some non-selected entities, then the shared points are not moved.

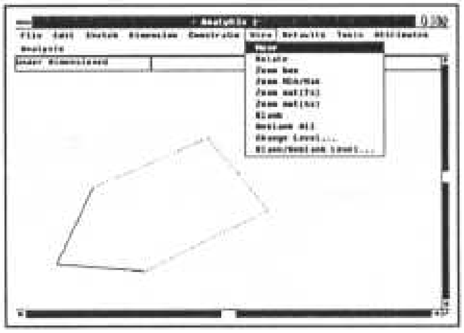
For example, if you select a single edge of a polygon, then perform a move, nothing will move as both vertices of the line are shared with non-selected edges.

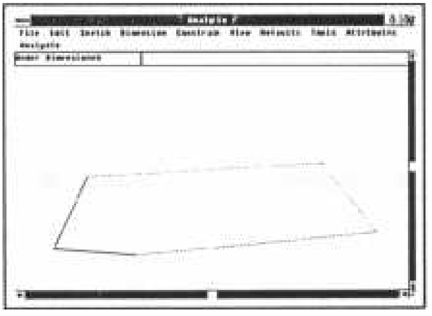
If you select two adjacent sides, then perform a move, the common point will move.





If you select three adjacent sides, both common points will move. This has the effect of moving the complete middle edge.





Notes & Alternatives

Note that the Move and Rotate operations are performed on the sketch, before the drawing is consistently dimensioned. If you have a consistently dimensioned drawing, the way to alter part of the drawing is by changing one or more dimensions.

However, if Analytix has adopted a different solution to your set of dimensions to the one you intended, you may wish to move some part of the drawing to give Analytix a better sketch to work from. The move, however, will not change the assigned dimensions in the drawing.

To change the position of a dimension relative to the geometry, Select-and-Drag it.

To move a single point Select-and-Drag it.

To move the whole drawing on the screen, use the scroll bars and View/Zoom functions.

View / Rotate

This lets you rotate the currently selected portion of the sketch.

To perform a rotation do the following:

1 - Select the geometrical objects to be rotated (Dimensions on their own are not rotated, but follow the geometry.)

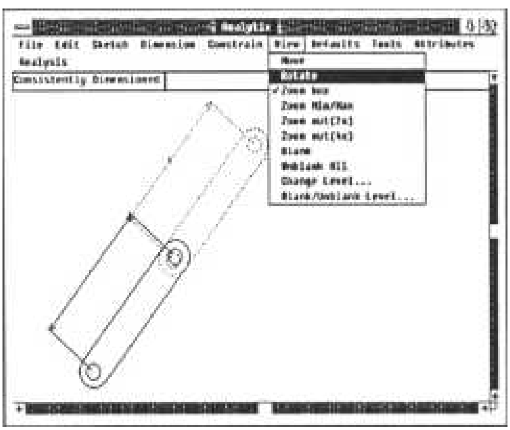
2 - Choose Rotate from the View menu.

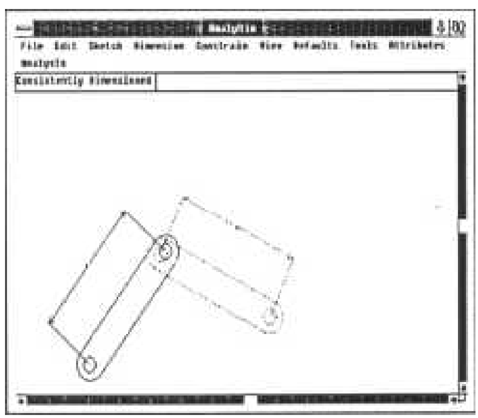
3 - Click on the center of the rotation (this may be any point on the screen.)

4 - Position the cursor over some part of the object to be rotated (away from the center of rotation).

5 - Press down the left mouse button and drag the object round.

6 - When the object is correctly positioned, release the mouse button.





What is rotated

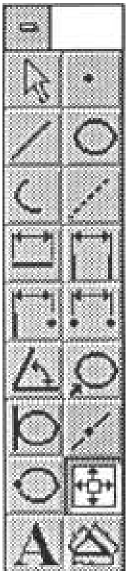
If your set of selected entities shares points with some non-selected entities, then the shared points are not rotated.

For example, if you select a single edge of a polygon, then perform a rotate, nothing will move, as both vertices of the line are shared with non-selected edges.

If you select two adjacent sides, then perform a rotate, the common point will be rotated about the center.

If you select three adjacent sides, both shared points will move, with the effect that the the common line will be seen to rotate about the center.

View / Zoom Box



This function lets you zoom in on your drawing.

To zoom in on the drawing:

1 - Choose the View / Zoom Box icon from the tool box.

2 - Position the cursor at the top left corner of the region which you wish to fill the entire window.

3 - Press down on the left mouse button and drag the mouse to the bottom right corner of the region, creating a rectangle delineating the region.

4 - Release the mouse button.

Analytix immediately zooms in on your region.

View / Zoom Max/Min

This menu option zooms to fit the drawing optimally to the screen.

View / Zoom Out (2x)

This option zooms out by a factor of 2.

View / Zoom Out (4x)

This option zooms out by a factor of 4.

Use the Zoom Out options followed by Zoom Box to achieve a finer Zoom Out.

View / Blank

Blanks the currently selected entities.

A blanked entity is not visible on the screen but it is still there. For example, a dimension still constrains the drawing when it is blanked.

Blanked entities are not selectable except by using the Select All options.

View / Unblank All

Unblanks all the currently blanked entities.

More selective control of which entities are visible and which are not may be achieved using the level management facilities described below.

View / Change Level...

All entities in Analytix are drawn on a specific "level". There are 16 different levels.

Levels may be blanked and unblanked individually, thus giving you a finer degree of control over what is visible than the ordinary Blank and Unblank functions.

By default everything is placed on the first level.

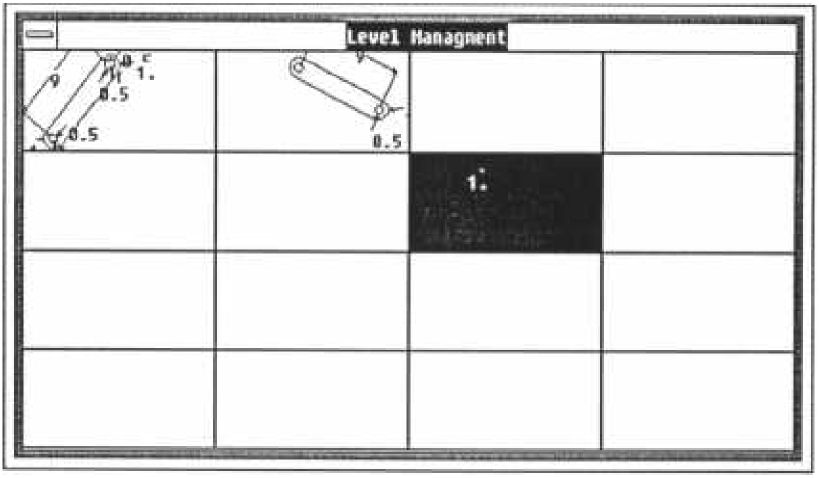
If there are no currently selected entities, View / Change Level changes the level where subsequent entities are placed.

If you have selected entities, these will be moved to the new level.

To change levels

1 - (Optionally) select entities whose level you wish to change.

2 - Choose Change Level from the View menu.



3 - The Level Selection Box appears. Click on the level you wish to change to.

4 - Close the Level Selection Box by double clicking on its System Menu Box in the top left corner.

The Level Selection Box contains sixteen miniature screen pictures, each containing the entities in that level. The current level is highlighted by displaying the reverse image.

To choose a new level, click on the miniature screen representing that level.

View/ Level Blank/Unblank...

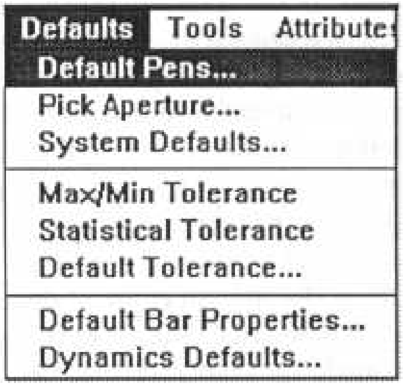
This option lets you select which levels are visible and which are not.

When you pick the Level Blank/Unblank menu option, you will see the Level Selection Box. The currently visible levels will be displayed normally, the currently blanked levels in reverse.

Click on the miniature screen representing a level to toggle it from blanked to unblanked or vice versa.

When you are finished, double click on the System Menu Button of the Level Selection Box to close the Box and cause the changes to go into effect.

Defaults



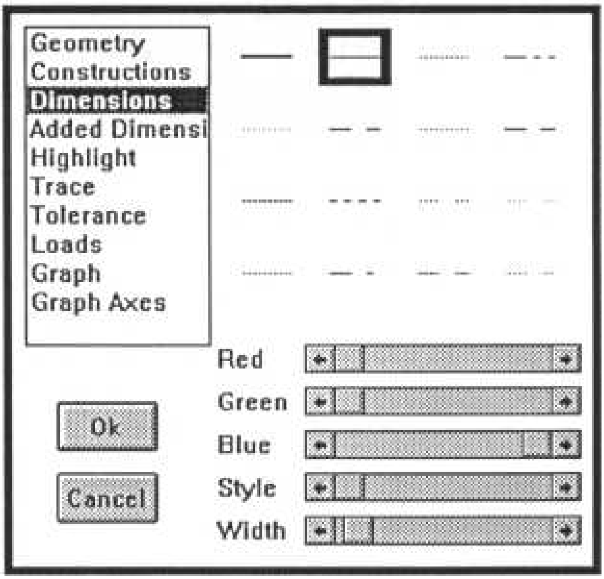
The Defaults menu lets you set up various defaults for the system It lets you define pen colors and styles, units, default tolerance and tolerance types, and gravitational constant.

Defaults / Default Pens...

This menu option lets you set the colors, styles (solid, dotted, dashed, etc.) and width of the lines used by Analytix for various parts of your drawing.

Analytix has sixteen logical pens. The Default Pens dialog box allows you to set which of these sixteen pens is used for each different type of entity. It also lets you set the color, style and width of each pen.

The List box in the top left corner of the dialog box lists the different types of entities which may be drawn. Sixteen buttons to the right of the dialog box show the current color, style and width of the different pens. Sliders at the bottom of the screen control the color, style and width of the current pen.



The different entities which may be given an individual pen are:

- Geometry - all lines (except construction lines), arcs and circles.
- Constructions - Construction lines.
- Dimensions - all dimensions created by the user.
- Added Dimensions - dimensions created by Analytix when Dimension / Automatic is invoked.
- Highlight - the pen used to highlight selected entities.
- Trace - the pen used to trace the path of a point in the Tools / Trace option.
- Tolerance - tolerance zones.
- Loads - external forces and moments added using the Analysis / Add Load menu option.
- Graph - the graph line.
- Graph axes.

When you click on one of these entities the current default pen for that entity is highlighted.

To change the default pen, position the cursor over one of the sixteen sample pens and click the mouse button.

To change the color, style or width of the current pen, use the Red Green and Blue slider bars to set red green and blue color values for the new color, use the style bar to set its style, and the width slider to adjust the width.

Note that changing the default pen for a given entity type means that, until the default is changed again, all objects of that type subsequently drawn will be drawn with the new pen. Existing objects will not be affected.

Changing the color and style of a given pen affects all objects in the drawing of that type.

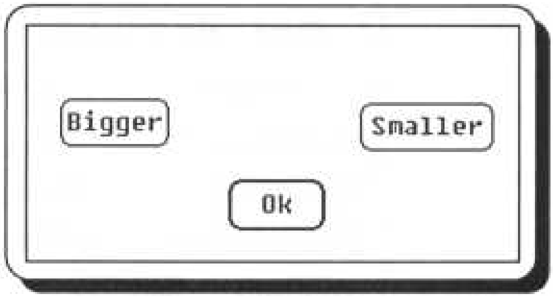
To change the pen on existing objects, use the Attributes / Pens menu option.

Defaults / Pick Aperture...

The pick aperture is a small box around the current cursor location which Analytix uses to decide when the cursor is close enough to be considered to lie on top of a given entity.

For example, when you are in Select mode, if you click the mouse button when a particular point lies within the pick aperture, then that point will be selected. If a line crosses through the pick aperture, but neither of its end points lie in the pick aperture, then that line will be selected.

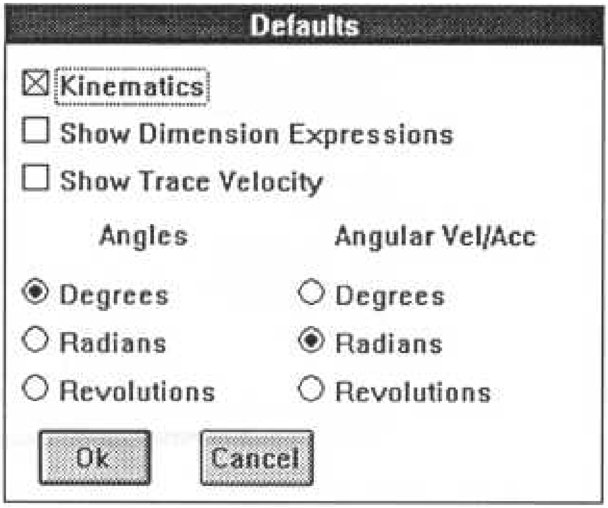
The Defaults / Pick Aperture menu option allows you to make the region bigger or smaller. Multiple clicks are meaningful.



Defaults / System Defaults

This menu option lets you specify certain settings.

- Whether kinematics is to be performed.
- Whether dimensions are to be shown as numerical values or as expressions.
- Whether the trace line style indicates velocity.
- Units for measuring angles
- Units for angular velocities and accelerations.



If Kinematics is enabled, Analytix calculates velocities and accelerations as well as positions for points and lines in the drawing. Turning Kinematics off will speed up the time it takes to make changes to the drawing. (Except in the animation tools, where Kinematics is automatically turned off.)

If Show Dimension Expressions is enabled, then any dimensions which are given as algebraic expressions will be displayed as expressions. If Show Dimension Expressions is not enabled, then the value of the expression will be displayed.

If Show Trace Velocity is enabled, the trace line will be broken so that each line segment starts at the position of the next step, thus displaying relative velocity of the trace at a glance. If Show Trace Velocity is not enabled, the trace line will be continuous.

As it is common practice for engineers to use different units for angles and angular velocities / accelerations. Analytix lets you set these units separately.

The initial default is for angles to be measured as degrees and for angular velocities and accelerations to be measured in radians / time and radians / (time).

Defaults / Max Min Tolerance

This option sets the tolerance analysis functions to return absolute tolerances, rather than statistical tolerances.

Tolerances input for dimensions are treated as absolute bounds, and the tolerances calculated for measurements from the drawing are absolute bounds on that measurement.

Tolerance zones calculated under max / min tolerancing represent the region inside which the point must lie.

Defaults / Statistical Tolerance

This option sets the tolerance analysis functions to return statistical tolerances rather than absolute tolerances.

The tolerances are set to come from Normal Distributions, which can be centered at the true value of the dimension, or may be centered away from the true position for asymmetric tolerance situations.

If U and L are the upper and lower tolerances and V is the stated value of the dimension, then the Normal Distribution has standard deviation points at V-L and V+U. That is, the actual value of the dimension is taken to come from a Normal Distribution mean V + (U-L)/2 and standard deviation (U+L) / 2.

For example if the stated value of the dimension is V=10.3, and the upper tolerance is 0.03 and the lower tolerance is 0.01, then we assume a Normal Distribution mean 10.31 and standard deviation 0.02.

Tolerance outputs for dimensions are standard deviation points of a Normal Distribution. If the upper and lower resultant tolerances are the same then the distribution is centered at the true value of the measurement. If the upper and lower tolerance values are not the same then the tolerance is asymmetrical and the Normal Distribution is off-centered.

Tolerance zones are the standard deviation equiprobable curve for the bivariate Normal Distribution of the position of the point of interest.

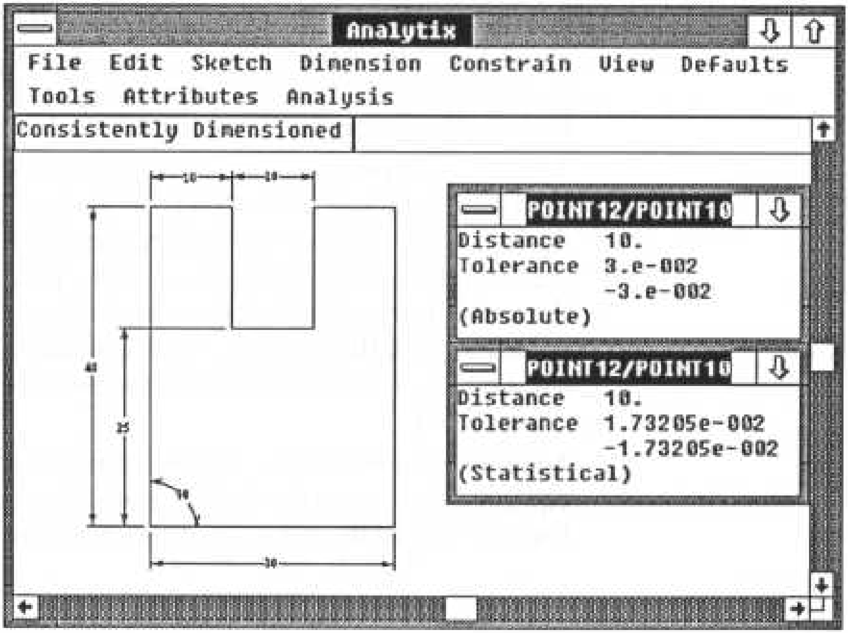
Note

Notice that, if you enter 3 standard deviation points rather than 1 standard deviation point for your input tolerances, then your output tolerances will be 3 standard deviation points.

Example

In the drawing shown, all the lengths have upper and lower tolerances of +0.01. We have measured the length of the undimensioned tab at the top right of the drawing. The upper measurement is under max / min tolerance conditions, the lower measurement under statistical tolerance conditions.

We see that under max / min conditions the tolerances stack up additively, under statistical conditions they stack up as the root sum squared.

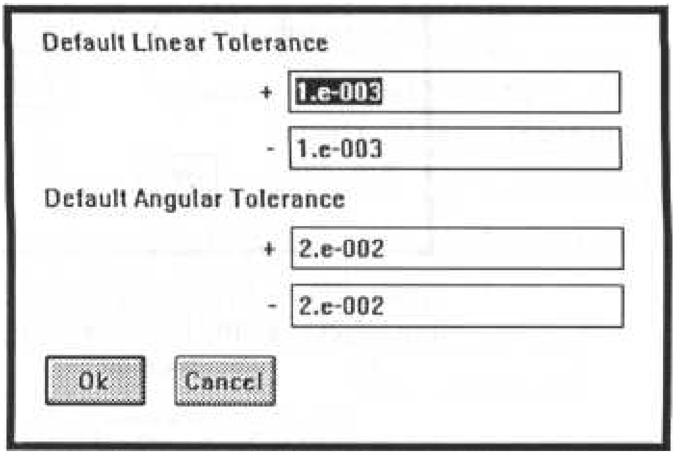


Defaults / Default Tolerances

This option lets you set default tolerances for all the linear and angular dimensions in your figure.

Tolerances may also be set individually by selecting a dimension and using the Attributes / Info menu option.

Setting the default tolerances causes all dimensions in the figure to have these tolerances. If you have already entered some special tolerances for specific dimensions, it will overwrite these. You should therefore use this option before setting specific tolerances using the dimension's Info box.

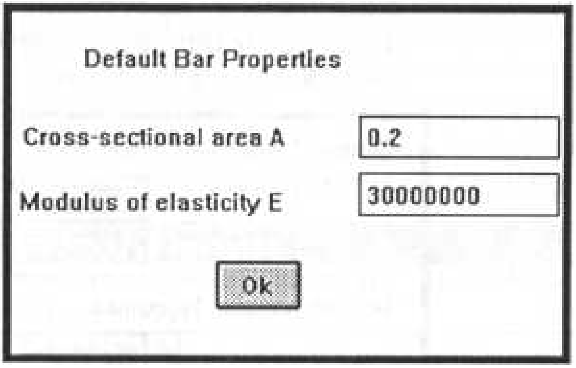


Defaults / Bar Properties...

Analytix allows you to do deflection and stress analysis on trusses.

In order to perform these analyses it is necessary to know the cross sectional area and modulus of elasticity of each bar in the truss.

The Defaults / Bar Properties menu option allows you to set default values for all the lines in the model.



To set different values for A and E for a particular line select that line then use Attributes / Info.

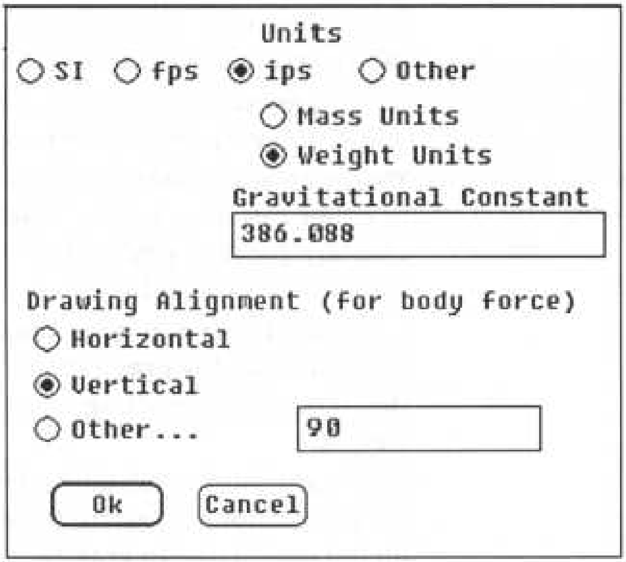
Defaults / Dynamics Defaults...

The Dynamics Defaults dialog box lets you specify three things:

1 - Whether your units use mass (inertial mass) or weight (force).

2 - The gravitational constant for your system of units.

3 - How your drawing is aligned with the gravitational force.



Standard Units

If you are using any of the standard unit systems included in Analytix: SI, fps, or ips, then you should click on the button corresponding to that system. Analytix will automatically set whether it is a Mass system or a Weight system, and will enter the appropriate gravitational constant.

Otherwise, click on the Other button, and you will need to set the either the Mass Units or the Weight Units button and enter in the Gravitational Constant.

Gravitational or Absolute Units

In Analytix, you may specify either the mass or the weight of objects. Which one you adopt will depend on whether you are using a gravitational system of units or an absolute system.

If you are using a gravitational system (such as fps or ips) you will probably wish to use weight rather than mass (pounds rather than slugs).

If you are using an absolute system (such as SI units) you will probably want to use mass rather than weight (kilograms rather than Newtons).

In either case you need to enter the gravitational constant for the system of units you are using. (That is the force exerted by gravity on a unit mass).

If you are using a gravitational system, then Analytix uses the gravitational constant to calculate the dynamic force acting on a given accelerating weight.

If you are using an absolute system, Analytix uses the gravitational constant to calculate the body force on the model due to gravity.

Drawing Alignment

In calculating resultant forces, Analytix considers three types of forces applied to the model.

1 - External applied forces and torques (Created using the Analysis / Add Load menu option).

2 - Dynamic forces due to accelerating masses and moments of inertia.

3 - Body forces due to the effect of gravity on masses.

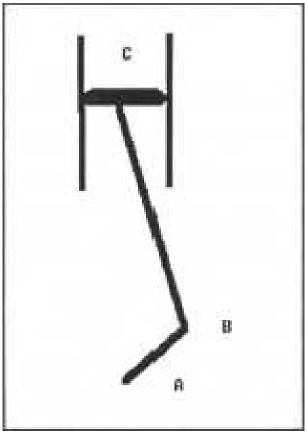
If your drawing is aligned horizontally then only the first two types of forces are considered.

If your drawing is aligned vertically, then each point has a gravitational force applied. The force is equivalent to the mass times the gravitational constant and is applied in the downward (negative) y direction.

If your drawing is inclined at some other angle theta, a force is applied which is equivalent to the mass times the gravitational constant times sin(theta). The force is applied in the negative y direction. This models the situation where the part is on a plane inclined at theta degrees to the horizontal.

Example

We wish to find the torque required to rotate crank AB at 120 rpm assuming the piston C has weight 0.51b, and neglecting the weight of AB and BC.



AB measures 2 inches and BC measures 8 inches.

We wish to solve the problem both for a horizontal orientation of the mechanism and for a vertical orientation.

First, as we wish to use revolutions as our unit of angular velocity, we need to change the default from radians. Do this using the System Defaults menu.

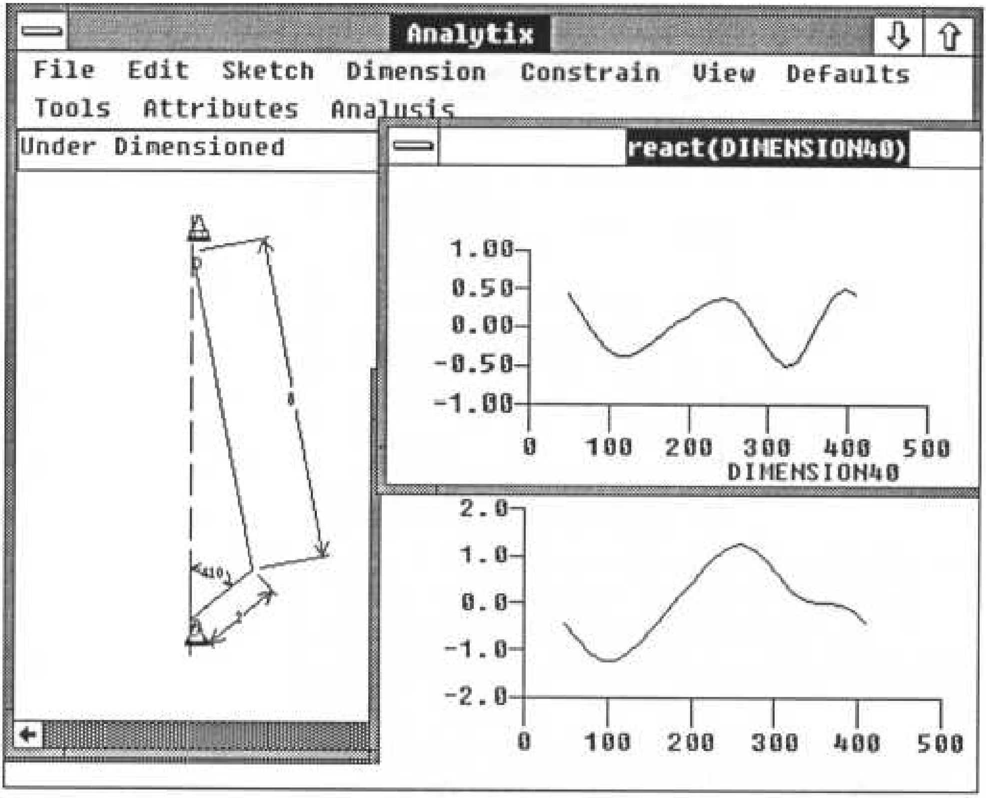
Now we wish to use inches, pounds and seconds. This is a gravitational system, hence we specify that we wish to use Weight Units.

We select ips as our unit system, Analytix automatically enters the correct gravitational constant and the fact that these are Weight Units.

We first specify a Horizontal, then a Vertical orientation.

The weight is entered as 0.5, the angular velocity as 2 revs per second. It is important to use revs per second rather than minute as the second is the basic time unit in ips. If you used minutes, then the gravitational constant (whose units are (distance)/ (time)^2) would have to be different.

The resultant torque in the angular dimension gives us the torque on crank AB.



Tools

This menu contains a number of tools to let you view the behavior of your model. There are drawing animation tools, calculation tools, drawing information tools and a switch to display the toolbox icon bar.

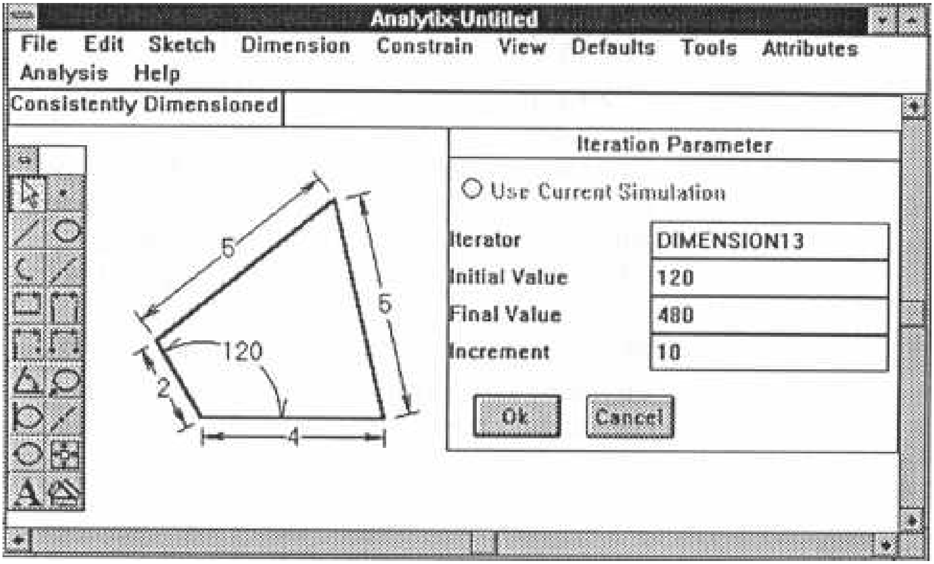
The animation tools: Animate, Increment, Trace and Envelope are grayed out if the drawing is not consistently dimensioned.

Tools / Animate

The Animate tool lets you watch your drawing move as Analytix steps the value of a variable through a prescribed range.

The variable will often be the value of some dimension.

When you pick Tools / Animate, you will see the Iteration Parameter Box. This allows you to either enter a new animation or use the current simulation if any (see Analysis / New Simulation later in this manual). To specify an animation, enter a variable for the parameter to be iterated on, the initial and final values for the parameter, and the required step size.



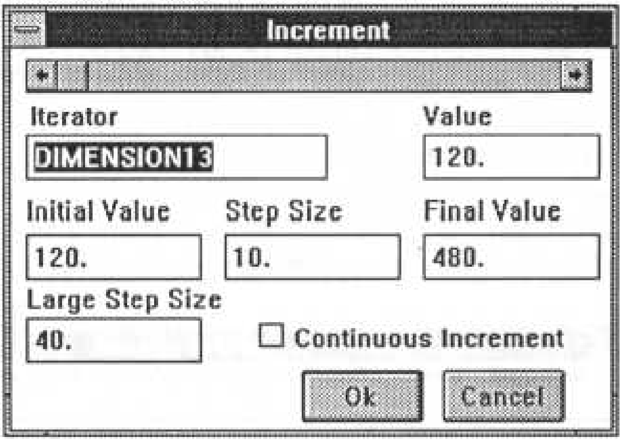
Frequently the iteration parameter will be one of the dimensions of the drawing. In this case, you can position the text cursor in the Iterator Box then go back into the main drawing and select the dimension. The dimension's name will be automatically entered into the Iterator Box.

When you have filled in the iteration parameters, click on Ok to start the animation.

The iteration parameter is remembered by Analytix. Next time you use Animate, Trace or Envelope you will notice that the parameter's details are already in the Iteration Parameter Box. To perform the same animation again, you just need to click on Ok.

Tools / Increment

This tool lets you step through an animation by incrementing (or decrementing) a particular variable. The variable would typically be a dimension or a variable on which one or more dimensions depend.



The edit control of the Increment box consists of: the variable to be incremented (Iterator), the current value of the Iterator (Value), its Initial Value, the quantity to be added or subtracted (Step Size), its Final Value, and a Large Step Size. If you select the Continuous Increment box, you will not be limited by the initial and final values.

To increment a dimension, make sure the text cursor is in the Iterator box, then return to the main Analytix window and select the dimension which you wish to alter. Alternatively, you may type a predefined variable directly into the Iterator box. Fill in the remaining edit control values.

Now use the arrows on the slider to increment the step size of the variable (or dimension) or drag the slider square along the bar to change the iterator's value. If you click the cursor on the slider bar in front of or behind the slider square the large step size is used. Each time the iterator value changes, the drawing is recalculated.

Close the Increment box by clicking on OK.

Tools / Trace

This tool draws the path followed by a point during the course of an animation.

To use the Trace tool, you must have:

1 - A consistently dimensioned drawing.

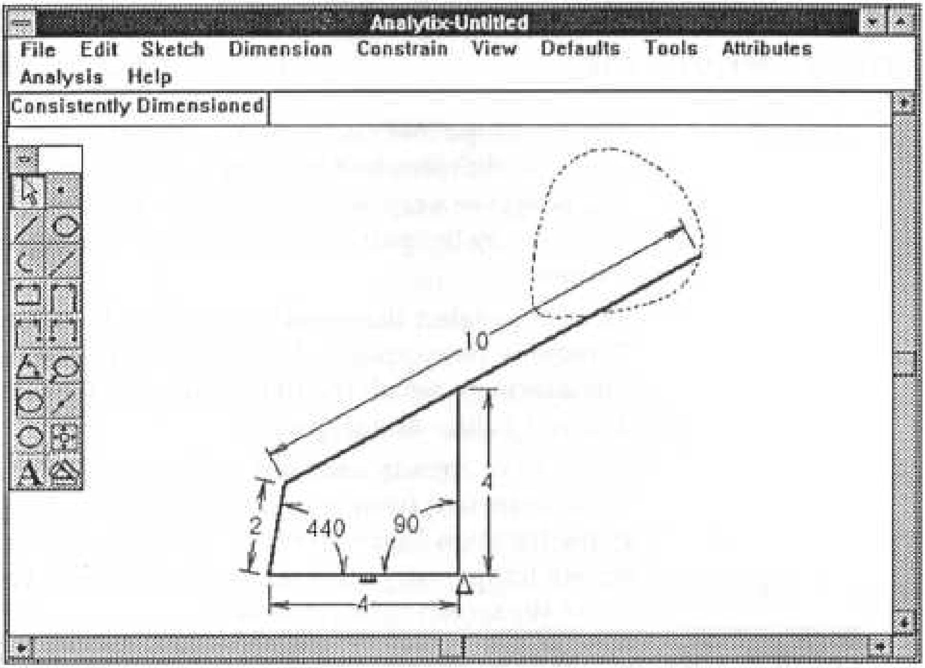
2 - One or more points selected.

Unless you have both of the above, the Trace menu option is grayed (disabled).

When you select the Trace tool, the Iteration Parameter Box appears. Fill in the variable or dimension on which the motion depends, its start and end values and step size. The Trace tool like the Animate tool can use the Current Simulation if one exists (see Analysis / New Simulation later in this manual).

If you have already used one of the animation tools, the information from your last use should already be in the Iteration Parameter Box.

The Trace Tool draws on a special layer of the drawing called the background layer. Traces cannot be individually selected and deleted. Instead you delete the whole background layer using the Edit / Erase Background menu option. The background layer is also used by the Tolerance Zone analysis tool.

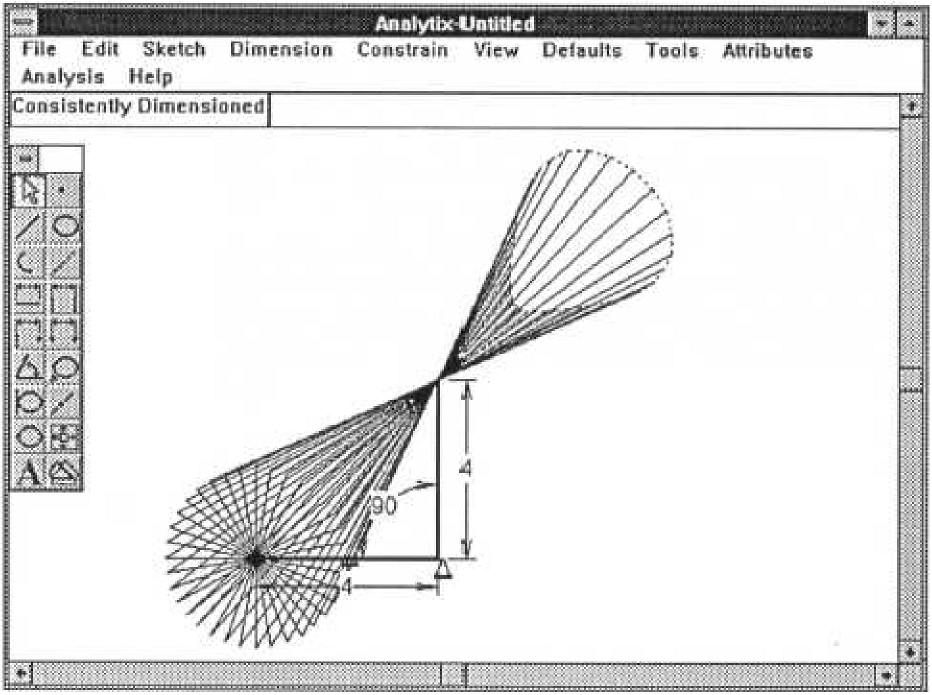

Analytix-Untitled
File Edit Sketch Dimension Constrain View Defaults Tools Attributes
Analysis Help
Consistently Dimensioned
10
4
2
440
90
4

Tools / Envelope

The Envelope tool performs an animation where the screen is not refreshed between frames. The effect of this is to give a representation of the total space occupied by the part over the prescribed range of motion.

When you select the envelope tool, the Iteration Parameter Box appears. Fill in the parameter or dimension on which the motion depends, its start and end values and step size.

If you have already used one of the animation tools, the information from your last use should already be in the Iteration Parameter Box. The envelope is only drawn temporarily. It is removed whenever you cause the screen to be refreshed.



Tools / Calculator

The Calculator is a tool which lets you evaluate expressions and define variables.

Expressions can be numeric:

- 5*(8-4.6)

or may contain variables:

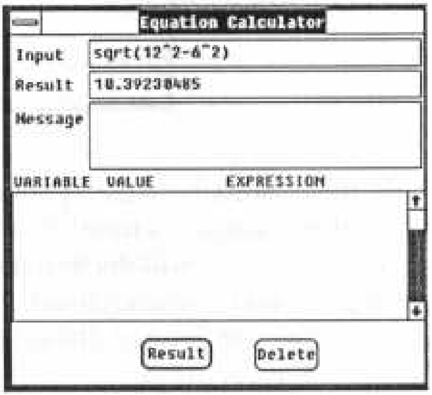
- a/(b**2-c**2)
- dist(point27,point13).

You can assign values to variables:

- a=27
- b= cos(45)
- DIMENSION17 = 10

or you can assign expressions to variables:

- a = (b-c)/(d+e)
- DIMENSION21 = sqrt(a*a+b*b)

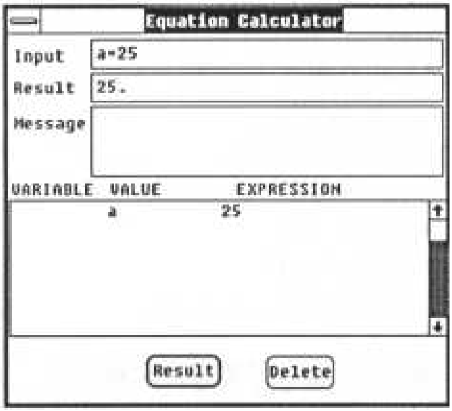


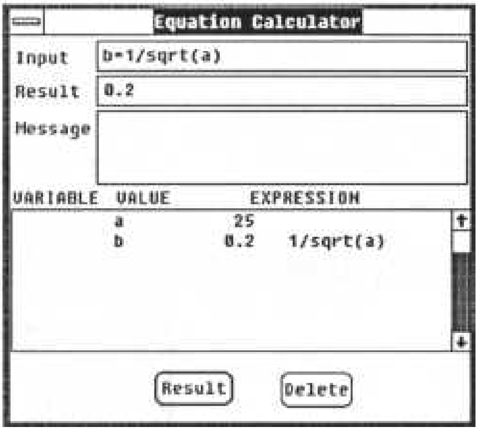
Variables (except for dimension names) must be at most 10 characters long and must start with a letter: e.g. rad , x12, bladevel

There is a special set of variables which are predefined by the drawing. These are the dimensions. The dimension name consists of DIMENSION followed by the number of that dimension.

e.g. DIMENSION17, DIMENSION237

If you assign a dimension variable, this has the effect of inserting the value or expression on the right hand side of the assignment into that dimension in





the drawing.

If the expression which you assign to a variable itself contains variables which have not been declared yet, the Calculator will declare the value of the expression to be undefined. When the necessary variables are defined, the calculator will fill in the values.

You cannot enter circular definitions: e.g.

a = b+c

c = 2*b-a

This is not allowed as c is in the expression which defines a and a is in the expression which defines c.

You can, however enter:

a = b+c

b=27

c = sqrt(b)

even though b and c are not defined at the time when a is declared.

Using the Calculator

To enter expressions:

Type the expression in the Input box. When the expression is entered, click on the Result button. The value of the expression will appear in the Result box.

If there is an error evaluating your expression, a message will appear in the Message area.

To assign variables:

Enter an assignment statement of the form

[variable name] = [expression]

in the Input box.

The variable will be entered into the table of variables displayed in the window at the bottom of the Calculator box.

If an expression (rather than a constant value) is given for a variable, both the expression and its current value are given

To edit a variable:

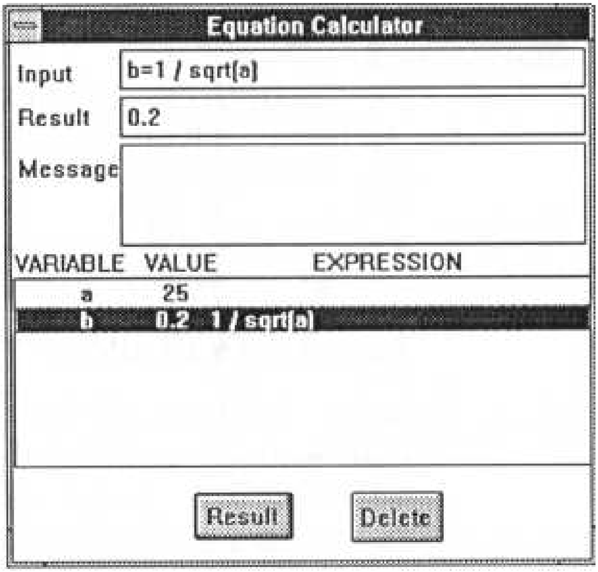
To edit a variable which has been entered into the variable table, click on that entry in the variable table.

The assignment statement for that variable will appear in the Input Box and the value of the variable will appear in the Result Box.

You can now go ahead and edit the assignment statement.

To delete variables

Delete a variable by highlighting it in the variables table then clicking on the Delete button.



A variable will not be deleted if other variables depend upon it. You will have to go and delete those dependent variable first.

Interacting with the drawing

The calculator interacts with the drawing in a number of ways:

1 - Dimension values may be assigned from within the calculator. e.g.
DIMENSION7=sqrt(2)

2 - Variable assignment statements may be incorporated in the definition of a dimension. (You might define the dimension value to be a=sqrt(2)).

3 - A number of Calculator functions take as arguments the names of points and lines, circles and dimensions in the figure. e.g. a=angle(LINE6,LINE8)

There are two ways to enter the name of a point, line, circle or dimension into the calculator. One is to type the name in. The second way is as follows:

1 - Position the text cursor in the input window at the place you wish the name to appear.

2 - *Go back to the main drawing and select the entity whose name you want. (You may have to move the Calculator out of the way first by dragging its title bar, or even move the drawing.)*

You will see the name of the entity appear in the Input box. If you wish the name to replace some of the existing text in the Input box, highlight this text before selecting the entity.

Valid Expressions

The arithmetic operations used in the calculator are:

sum

a+b

difference

a-b

product

a*b

division

a/b

power

a^b or a**b

parentheses

a-(b+c)

The logical operations used in the calculator are:

greater than

a>b

less than

a<b

greater than or equal to

a>=b

less than or equal to

a<=b

not equal to

a!=b

equal to (logical)

a==b

logical AND

aANDb

logical OR

aORb

logical NOT

a NOT b

Functions

Here we will list the different functions which are available in Analytix. These functions may be used as part of expressions, either in the calculator or in specifying what expression to graph or table.

Functions take as arguments one or more real numbers, or points, or lines or dimensions.

Functions will first be listed by category. They will then be listed alphabetically.

Functions By Category

Mathematical

- arccos(real1)
- arcsin(real1)
- arctan(real1)
- cos(real1)
- exp(real1)
- ln(reall)
- log(real1)
- log10(real1)
- sin (reall)
- sqrt(real1)
- step(real1)
- tan(real1)

Geometrical

- angle(line1, line2)
- distance(point1, point2)
- distance(point1, linel)
- length(line1)
- xcoord(pointl)
- ycoord(point1)

Logical

- if(condition, statement, statement)

Area properties

- area(group1)
- Imax(group1)
- Imin(group1)
- Ix(group1)
- Ixy(group1)
- Iy(group1)

- Iz(group1)
- xcentroid(group1)
- ycentroid(group1)

Velocities & Accelerations

- aangle(line1, line2)
- acc(point1)
- adistance(point1, point2)
- adistance(point1, linel)
- alength(line1)
- angacc(line1)
- angvel(line1)
- vangle(line1, line2)
- vdistance(point1, point2)
- vdistance(point1, linel)
- vel(point1)
- vlength(line1)
- xacc(point1)
- xvel(point1)
- yacc(point1)
- yvel(point1)

Forces, Torques, Bending Moments & Deflections

- react(dimension1)
- react(point1)
- xreact(point1)
- yreact(point1)
- react(actuator1)
- moment(point1,line1)
- shear(point1,line1)
- stress(line1)
- xdef(point1)
- ydef(point1)

Functions - Alphabetical

aangle(line1, line2)

This function returns the second derivative of the angle between two lines. The result is given in current angular acceleration units (by default radians / time)

acc(point1)

This function returns the magnitude of the acceleration of point1.

adistance(point1, point2)

This function returns the second derivative of the distance between point1 and point2.

adistance(point1, line1)

This function returns the second derivative of the perpendicular distance between point1 and line1.

alength(line1)

This function returns the second derivative of the length of line1. The result is given in current angular acceleration units (by default radians / time).

angacc(line1)

This function returns the angular acceleration of line1. The acceleration is given in current angular acceleration units (by default radians / time.)

angle(line1, line2)

This function returns the angle between line1 and line2. The angle is given in current angular units (by default degrees).

angvel(line1)

This function returns the angular velocity of line1. The velocity is given in current angular velocity units (by default radians/time).

arccos(real1)

This function returns the inverse cosine of real1 in the range between 0 and 180. The value of real1 must lie between -1 and 1.

arcsin(real1)

This function returns the inverse sine of real1 in the range between -90 and 90. The value of real1 must lie between -1 and 1.

arctan(real1)

This function returns the inverse tan of real1 in the range between -90 and 90.

area(group1)

This function returns the area enclosed by the profile defined by group1.

cos(real1)

Returns the cosine of angle real1. Real1 is measured in current angular units (by default degrees).

distance(point1, point2)

Returns the distance between point1 and point2.

distance(point1,line1)

Returns the perpendicular distance between point1 and line1.

exp(real1)

Returns e^(real1)

if(condition, statement1, statement2)

This function evaluates the condition and returns the value of statement1 if the condition is true, or the value of statement2 if the condition is false.

Imax(group 1)

This function returns the maximum area moment of inertia through the centroid of the object defined by group1.

Imin(group1)

This function returns the minimum area moment of inertia through the centroid of the object defined by group1.

Ix(group1)

This function returns the area moment of inertia about the x axis through the centroid of the object defined by group1.

Ixy(group1)

This function returns the area product of inertia of the object defined by *group1*.

Iy(group 1)

This function returns the area moment of inertia about the x axis through the centroid of the object defined by *group1*.

Iz(group1)

This function returns the area moment of inertia about the z axis through the centroid of the object defined by *group1*.

length(line1)

Returns the length of *line1*.

ln(real1)

Returns the natural logarithm of *real1*. *Real1* must be positive.

log(reall)

Returns the natural logarithm of *real1*. *Real1* must be positive.

log10(real1)

Returns the base 10 logarithm of *real1*. *Real1* must be positive.

moment(point1, line1)

This function returns the bending moment of *line1* measured at *point1*. *(point1* should lie on *line1)*.

react(actuator1)

This function returns the force in *actuator1*. (Or the torque if *actuator1 is* an angular actuator).

react(dimension1)

This function returns the reaction force or torque in *dimension1*. It is equivalent to selecting Resultant Force/Torque from the Analysis menu.

react(point1)

This function returns the magnitude of the reaction force in *point1*.

shear(point1,line1)

This function returns the shear force at *point1* on *line1*.

sin(real1)

Returns the sine of angle *real1*. *Real1 is* measured in current angular units (by default degrees).

sqrt(reall)

Returns the square root of *real1*. *Real1* must be positive.

step(reall)

Returns 0 if *real1<0;* returns 1 if *real1>=0*.

stress(line1)

This function returns the stress in *line1*. (Used in truss analysis.)

tan(real1)

Returns the tangent of angle *real1*. *Real1 is* measured in current angular units (by default degrees).

vangle(line1, line2)

This function returns the first derivative of the angle between two lines. The result is given in current angular velocity units (by default radians/time)

vdistance(point1, point2)

This function returns the first derivative of the distance between two points.

vdistance(point1, line1)

This function returns the first derivative of the perpendicular distance between *point1* and *line1*.

vel(point1)

This function returns the magnitude of the velocity of *point1*.

vlength(line1)

This function returns first derivative of the length of *line1*.

xacc(point1)

Returns the x component of the acceleration of *pointl.*

xcentroid(group1)

This function returns the x-component of the centroid of the area defined by groupl.

xcoord(point1)

Returns the x coordinate of *point1.*

xdef(point1)

This function returns the x-component of the deflection of *point1.* (Used in truss analysis.)

xreact(point1)

This function returns the x component of the reaction force vector at *point1.*

xdef(point1)

Returns the x component of the velocity of *point1.*

yacc(point1)

Returns the y component of the acceleration of *point1.*

ycentroid(group1)

This function returns the y-component of the centroid of the area defined by group1.

ycoord(point1)

Returns the y coordinate of *point1*.

ydef(point1)

This function returns the y-component of the deflection of *point1*. (Used in truss analysis.)

yreact(point1)

This function returns the x component of the reaction force vector at *point1*.

yvel(point1)

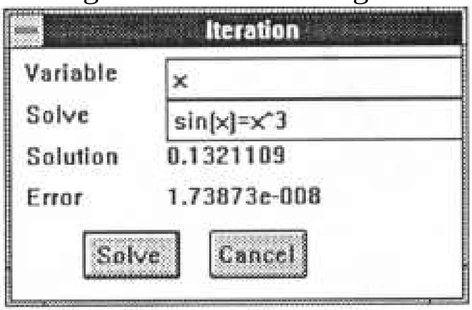
Returns the y component of the velocity of *point1*.

Tools / Univariate Iteration...

This tool implements a univariate root finder.

This uses an iterative technique to solve equations which cannot be written as a simple expression.

Enter the variable for which you wish to solve in the Variable box, and the equation you want solved in the Solve Box. Press Ok to calculate. The solution and error will appear below if the numerical analysis algorithms converge to a solution. Otherwise you will be given an error message.



While the Iteration tool is useful for solving equations, its real power comes when it is coupled with the drawing. This coupling is achieved when both the following are true:

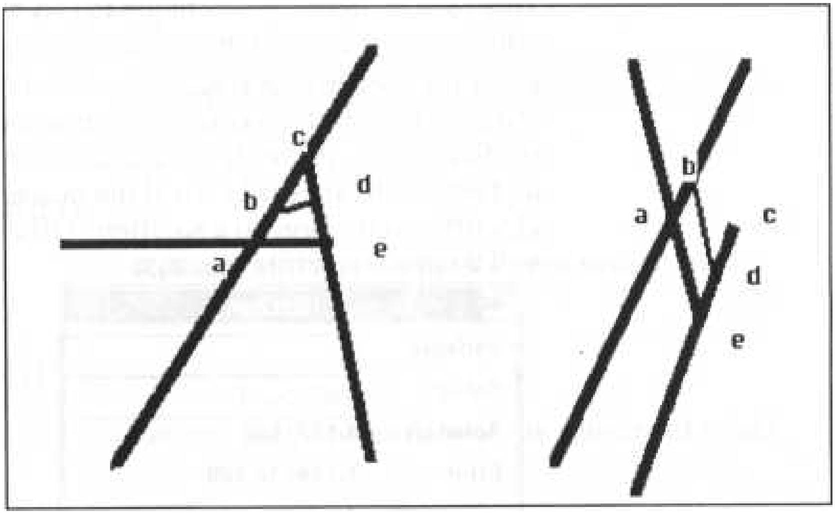
1 - The variable is an input to the drawing (either a dimension name or a variable on which one or more dimensions depend).

2 - The equation to be solved includes some functions which depend on the drawing (such as angles or distances).

We look below at an example of the use of the iterator in such a circumstance.

When drawing entity names (lines, points, circles, dimensions) are required in the Iteration Box they may be acquired by positioning the text cursor in the appropriate entry field then selecting the entity in the main drawing.

Example



The collapsible chair shown above should have angle bae 65 degrees. Given that ae is to be length 4, ce to be length 7, and bd length 3, find positions for b and d such that the chair is able to fold flat:

i.e. ab+ae = bd+de.

Enter a drawing of the triangle ace in the upright (unfolded) position of the chair.

Enter the given dimensions and a guess at the distance between a and b (2 is the guess used in the figure).

Now open the Iteration tool. Position the text cursor in the Variable Box, and select the dimension giving the distance between a and b.

In our case this is called DIMENSION15. (In your case it will probably have a different number.)

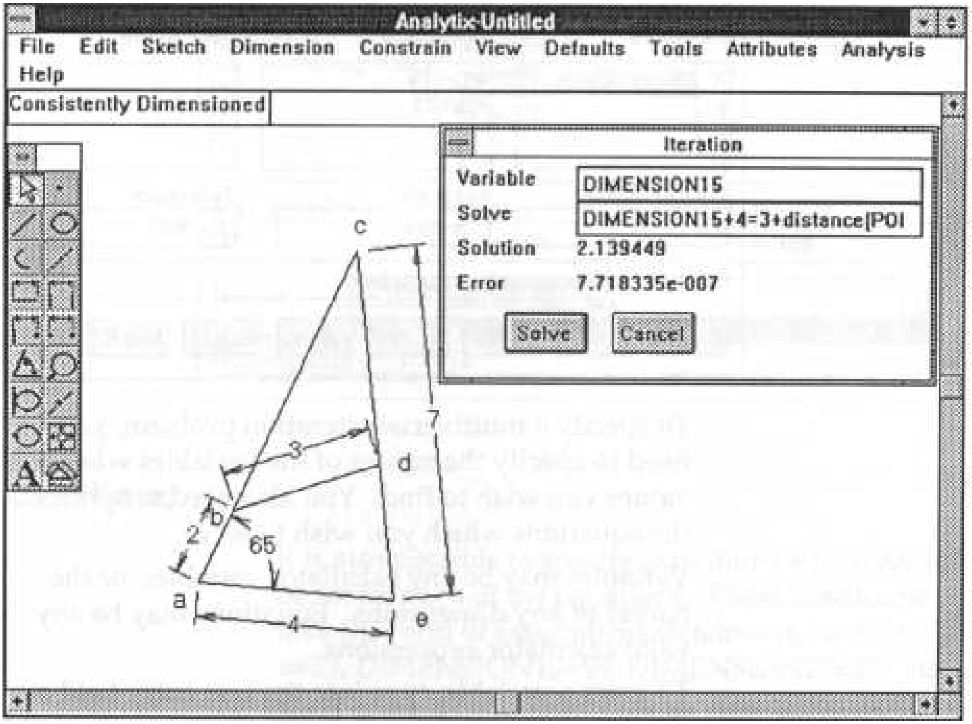
Now in the Solve Box, enter the equation

DIMENSION15+4=3+DISTANCE(POINT4,POINT9)

Except rather than POINT4 and POINT9 enter the names of points d and e in your diagram. (Do this by positioning the cursor at the correct position in the Solve Box then selecting the points.)

Press Solve to derive a solution to the problem.

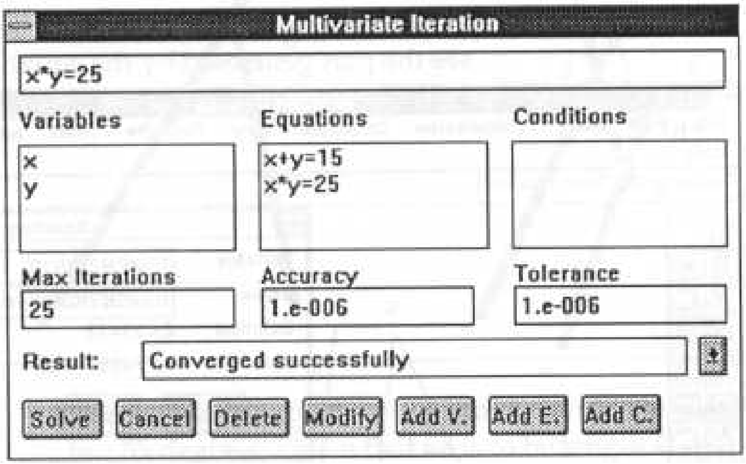
Now if you cause the screen to be repainted, you will see the part generated by the equation solver.



Tools / Multivariate Iteration

This tool implements a multivariate root finder.

it allows you to simultaneously solve a system of equations in a number of unknowns. (The number of equations must be equal to the number of variables.)



To specify a multivariate iteration problem, you need to specify the names of the variables whose values you wish to find. You also need to specify the equations which you wish to solve.

Variables may be any calculator variables, or the names of any dimensions. Equations may be any valid calculator expressions.

To enter a variable, type into the text entry field at the top of the multivariate iteration dialog, then click the Add V. button.

To enter an equation, type into the text entry field at the top of the multivariate iteration dialog, then click the Add E. button.

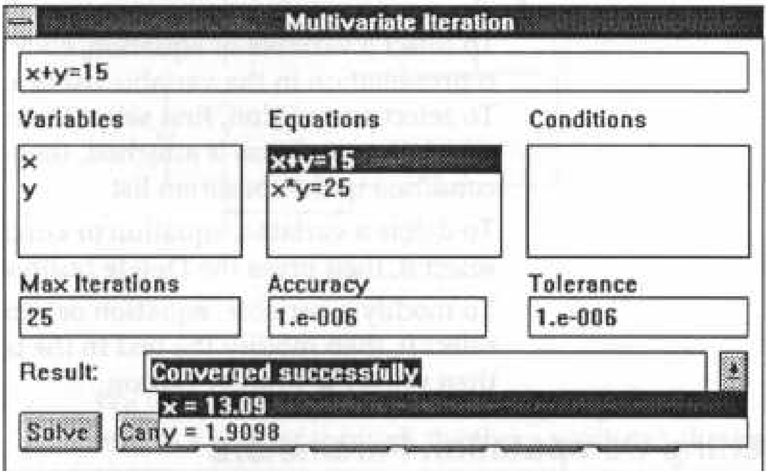
The example above shows the dialog box for solving the system

x+y=15;

x*y=25;

for x and y.

The values for x and y which the solver found may be inspected by clicking on the down arrow at the right of the Result box.



Conditions

It is also possible to specify conditions which are to be set for each of the equations. These conditions take the form of assignment statements, such as x=23, DIMENSION13=90, DIMENSION27=2*t, etc.

A set of such conditions may be attached to each equation. Before it tests the value of the equation, the multivariate root finder will first perform all the condition assignments attached to that equation.

The condition assignments are particularly useful for mechanism synthesis problems. Our second example problem (below) covers their use.

To enter a condition, first select the equation it is to be attached to. Then type into the text entry field at the top of the multivariate iteration dialog, then click the Add C. button.

Delete & Modify

To select a variable or equation, click on its representation in the variable list or the equation list. To select a condition, first select the equation to which the condition is attached, then click on the condition in the condition list.

To delete a variable, equation or condition, first select it, then press the Delete button.

To modify a variable, equation or condition, first select it, then modify the text in the text entry field, then press the Modify button.

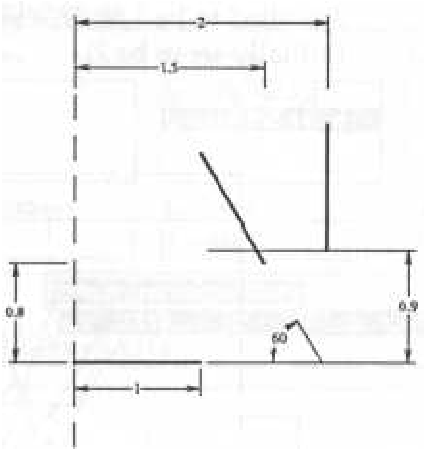
Setting Computation Parameters

You can set three parameters which let the iterative equation solver decide when to accept a solution, and when to give up trying to find a solution.

The maximum number of iterations specifies how many iterations of a Newton Raphson root finding algorithm will be performed before the solver gives up and returns a Numerical Error.

The Accuracy and Tolerance specify how close you need your solution to lie to the real solution. The solver accepts a solution as successfully converged if each variable is within the accuracy value of a true solution and the error in each equation is less than the tolerance value.

Example 1



We wish to synthesize a four bar linkage which moves a shutter through the three positions shown above.

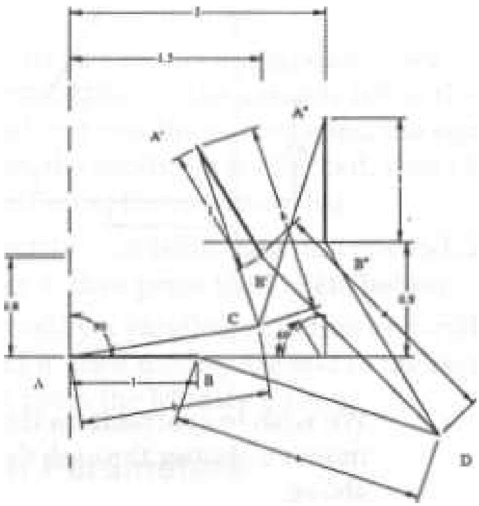
One way to do this is to create a single Analytix model which incorporates all three configurations (see below).

We draw a link from point A to another point C and a second line from A' to C representing the same link in the second configuration of the mechanism. We draw a third line from A'' to C representing the same link in the third configuration.

Similarly, we draw three lines BD B'D and B''D representing the three configurations of the link attached to B.

For this mechanism to work, we need the lengths of AC, A'C and A"C to be equal and we need the lengths BD B'D and B"D to be equal.

We specify AC and A'C to have length b (initially specified to be 1.5). We specify BC and B'C to be a (initially set to be 2).



Having set values for AC, and A'C, however, we are not able to specify a value for A"C, and this is not 1.5. Hence we need to find the value of a which makes the distance A"C equal to b. We also need to find the value of b which makes B"D equal to a.

We can do this using the multivariate iteration tool.

The variables which we wish to solve for are a and b.

The equations which we wish to solve are:

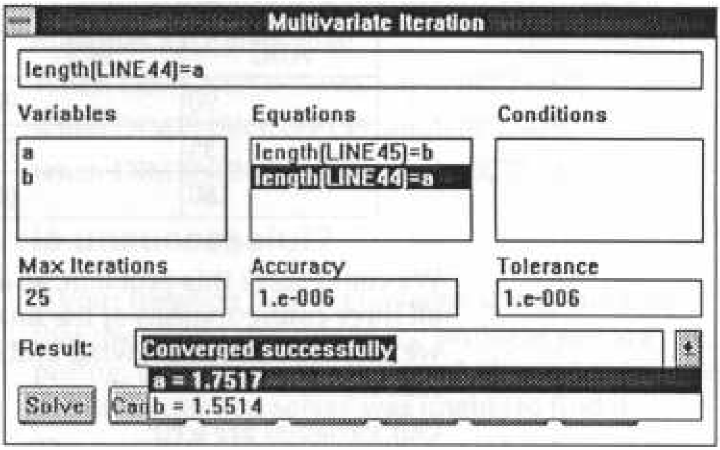
length(A"C)=b

length(B"D)=a

Solving finds a solution

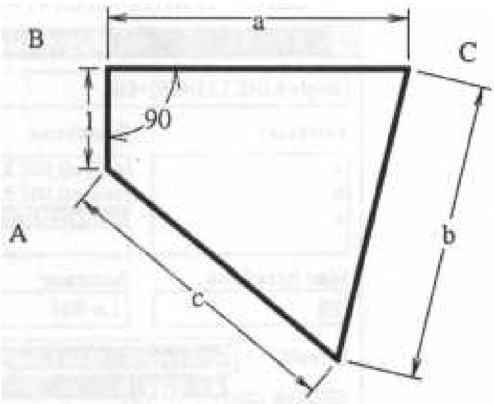
a=1.7517

b=1.5514



Example 2

In our second example, we show how conditions can be used to facilitate the solution of certain types of synthesis problems.



We wish to synthesize a four bar linkage where the output link angle BCD has the following relationship to the input link angle ABC:

ABC	BCD
90	90
135	75
180	60

We could solve this problem, as above, by drawing all three configurations of the linkage. Alternatively we can use a single drawing and set conditions as follows:

Our variables are a,b,c.

Our equations are:

angle(BC,CD)=90

angle(BC,CD)=75

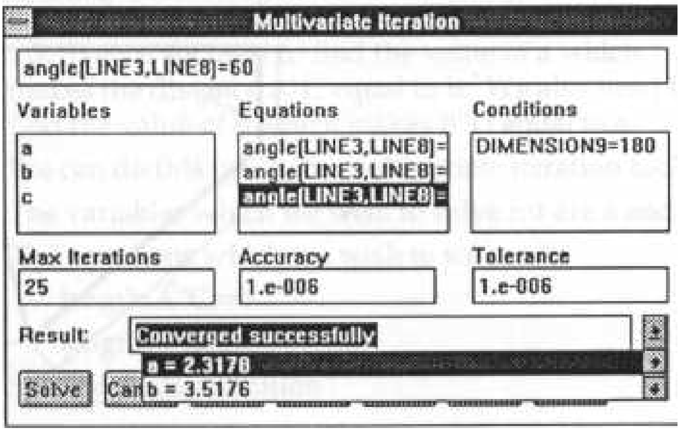
angle(BC,CD)=60

We have one condition on each equation:

(i) DIMENSION9=90

(ii) DIMENSION9=135

(iii) DIMENSION9=180



DIMENSION9 is the angle dimension between AB and BC.

The multivariate iteration tool will now look for a set of values a,b,c such that:

When DIMENSION9=90, angle BCD = 90,

when DIMENSION9=135, angleBCD=75,

when DIMENSION9=180, angle BCD =60.

What if iteration is unsuccessful?

If your iteration fails to converge, it may indicate that there is no solution to the problem you are posing, or it may mean that a solution exists but the iterative equation solver was unable to find it.

The equation solver uses the current values of the input variables as a "first guess" at the solution it may be that by changing the initial values of the variables you can help the equation solver converge.

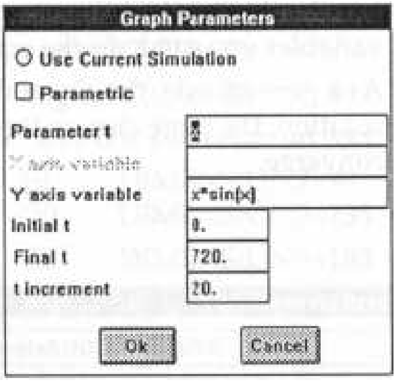
As a general rule, the closer the initial guess is to the solution, the more chance there is that the solver will converge.

Tools / Graph...

This tool lets you create a graph.

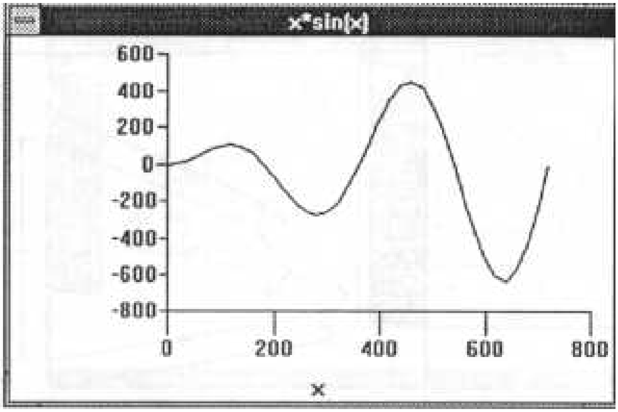
When you select the Graph tool you are presented with a dialog box with some choices. If you have built a simulation (see Analysis / New Simulation later in this manual), the Use Current Simulation button will be selected and you can autimatically display the last simulation created. If you deselect the Use Current Simulation or have not created a simulation, you have a choice between a graph or a parametric plot.

In the case of a simple graph, you to enter a variable for parameter t, an expression for the y axis, and an initial value, final value and step size for t.



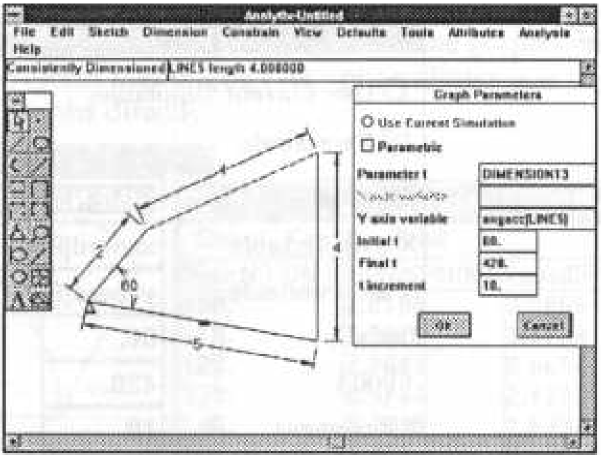
If you have already used one of the animation tools, then the values which you used for your iteration parameter appear as defaults in the Parameter t, Initial t, Final t and t increment boxes.

Click on Ok to see the graph.

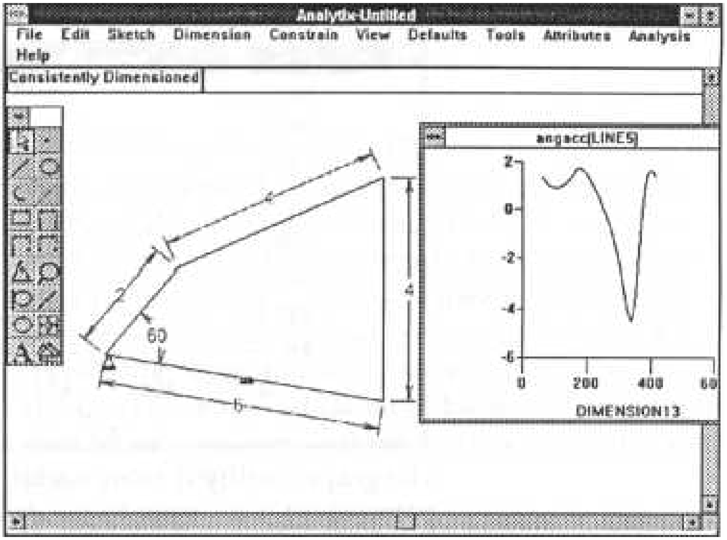


The graph facility is most useful, however, when the parameter t is an input to the drawing and the y-axis variable is an output.

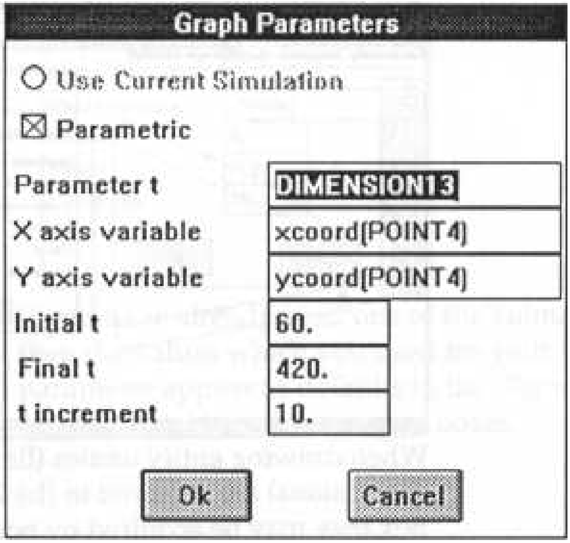
For example, if the parameter t is the value of a particular dimension, and the y-axis variable is the angular acceleration of a particular line.



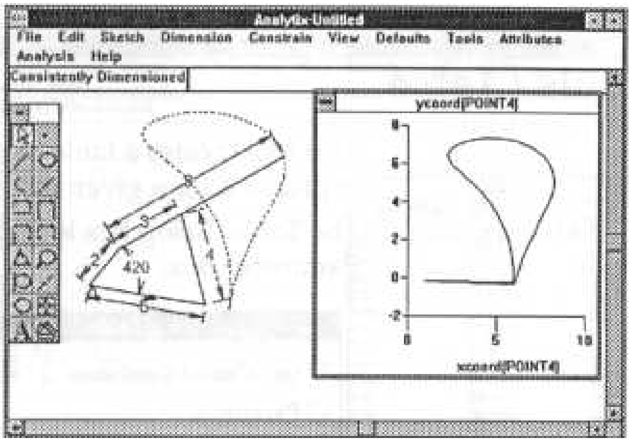
When drawing entity names (lines, points, circles, dimensions) are required in the Graph Parameters Box they may be acquired by positioning the text cursor in the appropriate entry field then selecting the entity in the main drawing.



A trace or displacement curve is a simple example of a parametric plot. When Parametric is selected the x axis variable is added to the graph parameters which you specify.



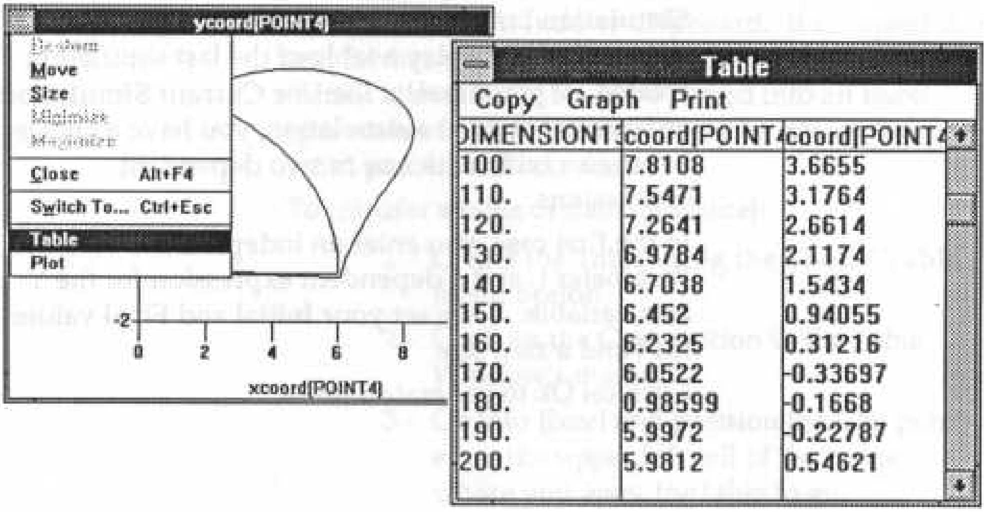
To remove the graph from the screen, double click in



its System Menu Box in the upper left corner of the graph window.

Graph Window's System Menu

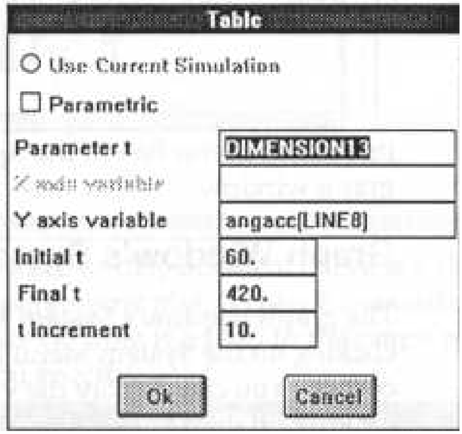
The graph window's System Menu is obtained by clicking on the System Menu Box in it's upper left comer. You can display the values as a table by clicking on the Table option of the graph window's System Menu. The Plot option allows you to plot graphs directly.



Tools / Table...

This tool creates a table of the values of an expression for a given range of an expression.

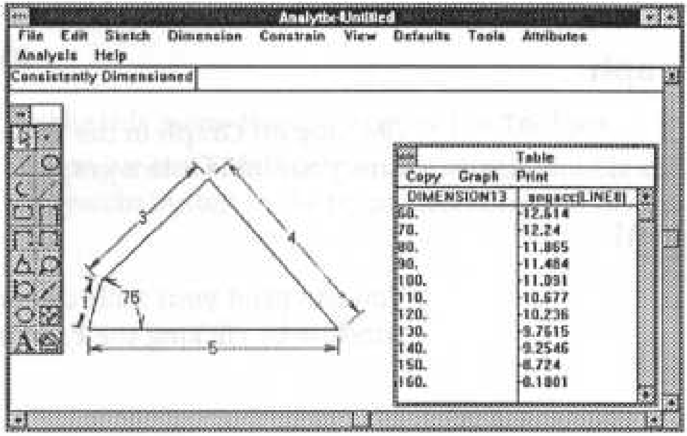
The Table dialog box is identical to the Graph Parameter Box.



If you have built a simulation (see Analysis / New Simulation later in this manual), the Use Current Simulation button will be selected and you can automatically display a table of the last simulation created. If you deselect the Use Current Simulation or have not created a simulation, you have a choice between a table with one or two dependent expressions.

In the first case, you enter an independent variable, Parameter t, and a dependent expression for the Y axis variable. Then set your Initial and Final values for t and a step size.

Click on Ok to generate a table.



The table originally appears small and in the top left of the screen. It can be repositioned by dragging its title bar and resized by dragging the frame around the Table window.

The Scroll bar in the Table allows you to scroll through the values.

Copy

The Copy option in the Table menu copies the table values onto the Windows Clipboard. It is copied in a text format with values separated by tabs. This format allows the table to be pasted into an Excel spreadsheet. Alternatively it may be pasted into a word processor document.

To transfer a table of data into Excel:

1 - Create the Table using the Tools / Table menu option.

2 - Click on the Copy option in the Table Window's menu.

3 - Go into Excel and position the Cell pointer on the upper left cell of the range where you want the table to go.

4 - Select Edit / Paste from the Excel menu.

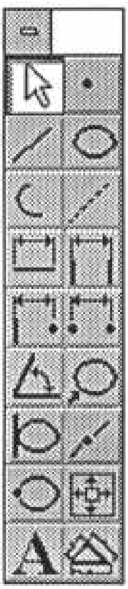
Graph

Clicking on Graph in the Table menu allows you to turn your table into a graph.

Print

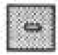
You can print your table directly from the Table window by clicking the Print button.

Tools / Toolbox

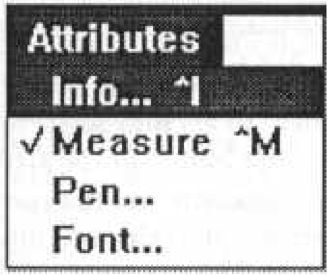


Invoke this menu item to display the Toolbox.

To remove the Toolbox from the screen, double click the system button in the upper left corner of the box.



Attributes



The Attributes menu contains two very important functions for communicating with the drawing.

1 - The Info function gives you information on the currently selected entity, and lets you add more information to it, such as the velocity and tolerance of a dimension, the mass of a point, the moment of inertia of a line.

2 - The Measure function lets you measure distances and angles (along with their tolerances) from the drawing.

There is also the Pen function which lets you change the style and color of the selected graphic entities, and the Font function lets you change the style and size of any alpha-numerics in the drawing.

Attributes / Info...

This function brings up the Info Box for the particular entity which is selected.

The Attributes / Info menu option is grayed out unless you have a single entity selected.

A shortcut to get into Select mode is to click on the right mouse button or the arrow icon in the Toolbox.

The Info box tells you the various parameters and properties of the selected entity, and lets you edit whichever ones are appropriate.

Keyboard Shortcut:

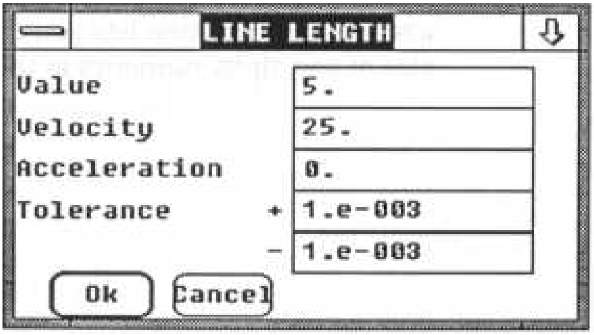
Press [Control] + I

Mouse Shortcut:

Double clicking on an entity while in select mode will select the entity and its Attributes / Info box.

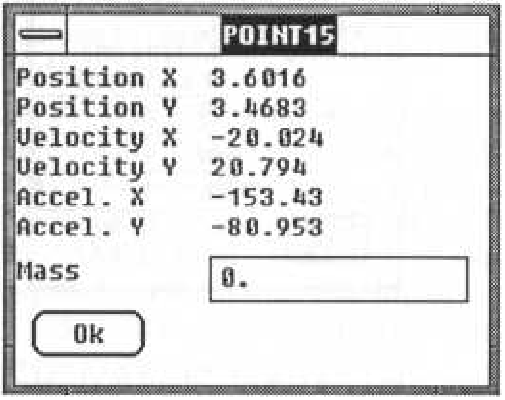
Dimension

The Dimension Info Box lets you edit the dimension's value, its velocity and acceleration and its tolerances.



Point

If you select a point then pick Attributes / Info, you will see the Points Info Box.



This tells you the position, velocity and acceleration of the point. It also lets you edit the mass of the point.

If you leave the Point Info Box on the screen while you change the value of a dimension using the Dimension Info Box, you will see the Point Info Box automatically update its value.

You can make the Point Info Box less wasteful of screen space by moving and resizing it.

Line

The Line Info Box tells you the length of a line, its angular velocity and acceleration, and tolerances on its length.

You can edit the moment of inertia of the line.

If the line represents the bar of a truss, you can define its cross sectional area (A) and its modulus of elasticity (E). (See also Defaults / Default Bar

Properties, and Analysis / Stress covered elsewhere in this manual.)

Actuator

In select mode, double clicking on an actuator symbol (or clicking on the Actuator then selecting Attributes / Info) brings up the Actuator dialog box. You can use this box to change the type of the actuator or to change it's parameters.

Force

In select mode, double clicking on an applied force (or clicking on the force then selecting Attributes / Info) brings up the Force Dialog Box. You can use this box to change the values of the applied force.

Expressions may be entered for force values. Forces are evaluated after geometry in Analytix. Hence the expression for a force may include functions of the geometry (such as ANGLE, DISTANCE, XVEL, ACC etc.). However an expression for a force should not involve a function which itself is dependent on the forces in the system (such as REACT).

Torque

In select mode, double clicking on an applied torque (or clicking on the torque then selecting Attributes / Info) brings up the Torque Dialog Box. You can use this box to change the value of the applied torque or its direction.

Expressions may be entered for torque values. Torques are evaluated after geometry in Analytix. Hence the expression for a torque may include functions of the geometry (such as ANGLE, DISTANCE, XVEL, ACC etc.). However an expression for a torque should not involve a function which itself is dependent on the forces and torques in the system (such as REACT).

Group

Double clicking on a group while in select mode (or clicking on the group then selecting Attributes / Info) brings up a box which displays the area mass properties of the group.

These properties are:

Area - the area of the outer profile minus any profiles contained within. (If the profiles intersect the result is meaningless.)

Ix - Area moment about the x-axis through the centroid.

Iy - Area moment about the y-axis through the centroid.

Iz - Area moment about the z-axis through the centroid. (This is sometimes denoted J).

Ixy - Area product of inertia through the centroid.

Attributes / Measure

This function lets you measure quantities from the drawing.

You can measure:

- The distance between two points.
- The perpendicular distance between a point and a line.
  The angle between two lines.

Relative velocities (first derivatives) and accelerations (second derivatives) of these measurements are also reported.

Tolerances as well as tolerance sensitivities are given for these measurements.

The Measure box tells you whether the tolerances were measured under an assumption of absolute or statistical tolerance accumulation.

If you leave the Measure Box on screen, you can change dimension values and watch the measurement and other reported values update.

To remove the Measure Box, double click on its System Menu Box in its top left comer.

Distance between two points

To measure the distance between two points:

1 - Select the first point.

2 - Holding down the [Shift] key, select the second point.

3 - Pick Attributes/ Measure (^M).

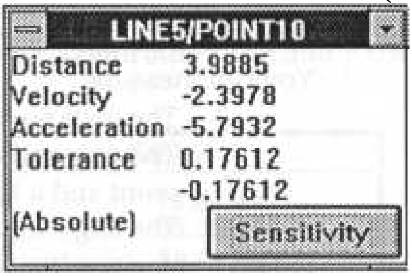
Distance between a point and a line

To measure the distance between a point and a line:

1 - Select the point (or the line).

2 - Holding down the [Shift] key, select the line (or the point).

3 - Pick Attributes / Measure (^M).



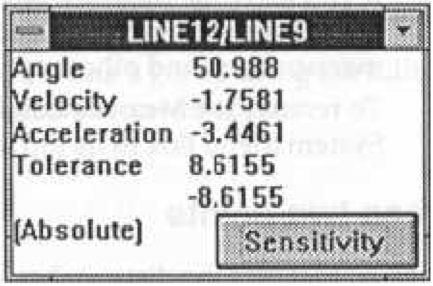
Angle between two lines

To measure the angle between two lines:

1 - Select the first line.

2 - Holding down the [Shift] key, select the second line.

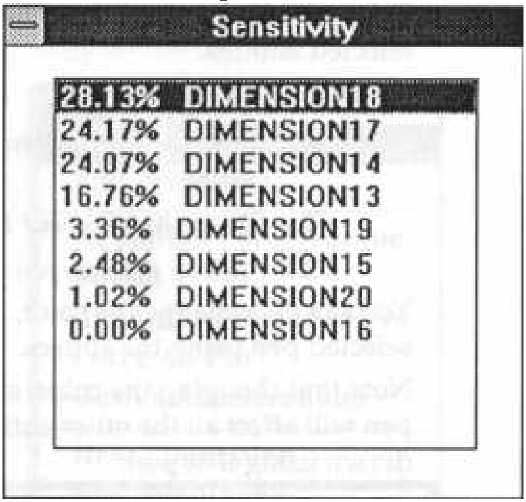
3 - Pick Attributes / Measure (^M).



Tolerance Sensitivity Analysis

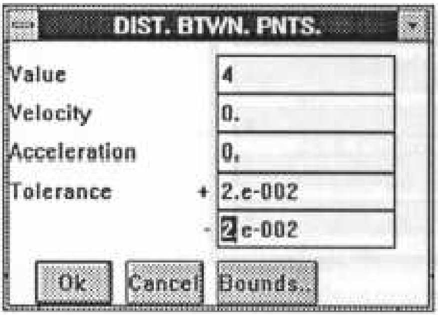
Click on the Sensitivity button in the Measure Box to display the effective tolerance on a distance or angle measured from the drawing.

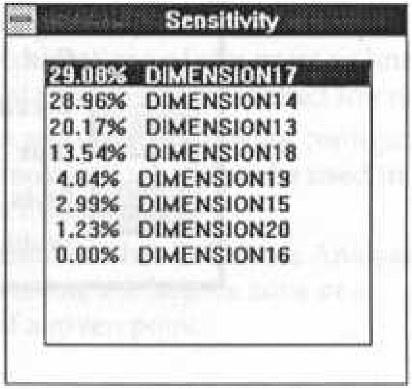
You see the Tolerance Sensitivity display. This lists the dimensions in the figure along with the percentage contribution of each dimension to the tolerance stack up.



If you click on a particular dimension name in the tolerance sensitivity box, that dimension will be highlighted in the drawing.

If you double click on the dimension name, the Info Box for that dimension is displayed. This enables you to adjust the tolerance on the selected dimension and watch the effect of the adjustment on its contribution to the stack-up.





Attributes / Pen...

This option lets you change the pen used for the selected entities.

To use this function:

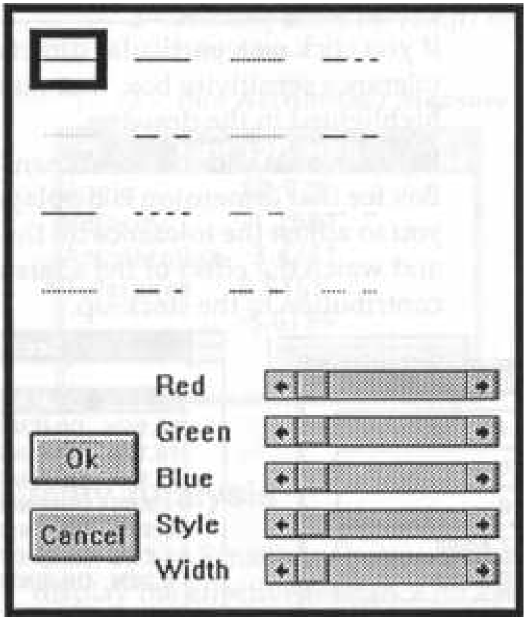
1 - Select the entities whose pen you wish to change.

2 - Choose Attributes / Pen.

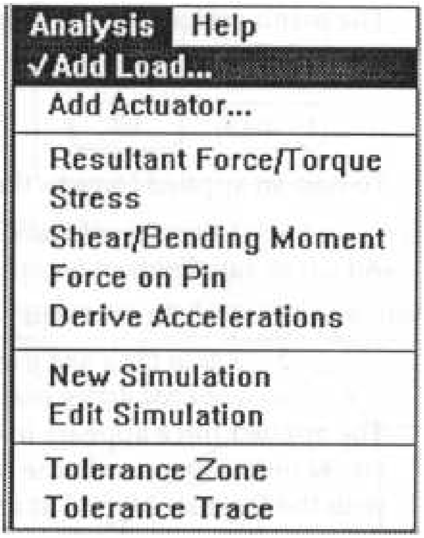
3 - Click on the new pen you wish to use.

You can also change the color, style and width of the selected pen using the sliders.

Note that changing the color, style and width of a pen will affect all the other entities which have been drawn using that pen.



Analysis



The Analysis menu contains options to allow you to add external forces and torques and actuators to the drawing, to measure the resultant force in a dimension or on a point, to analyze stress and deflection of trusses and shear force and bending moments at some point on a line, and to report instantaneous accelerations of any point or line in a model which contains masses or applied forces.

You can collect a sequence of drawing configurations as a playback simulation which can be used in many of the Tools functions.

The Analysis menu also has Tolerance Analysis functions which create a tolerance zone or a tolerance trace of a given point.

Analysis / Add Load...

This is the option which allows you to add an externally applied force or torque to the drawing.

The menu option is grayed out unless a point or line is selected.

Applied Force

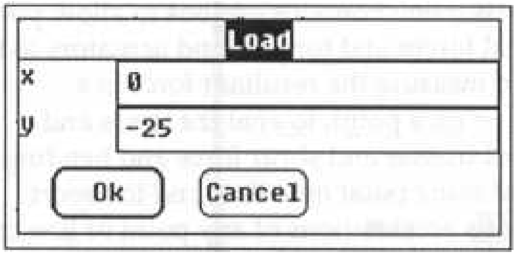
To add an applied force to the model:

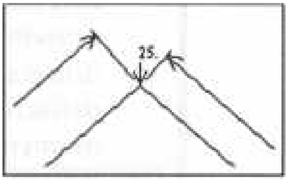
1 - Select the point where the force is to be applied.

2 - Pick the Analysis l Add Load menu option.

3 - Fill in the x and y components of the force in the Load dialog box.

The applied force appears in the diagram as a small arrow pointing towards the point of application, with the force size typed at its tail.





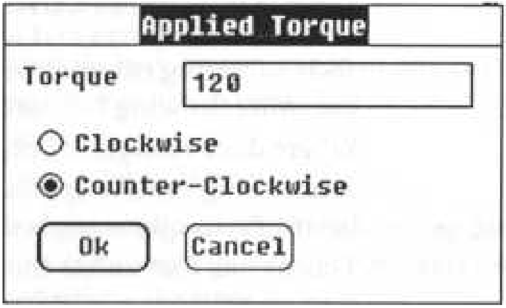
Applied Torque

To add an externally applied torque to the model:

1 - Select the line to which the torque is to be applied.

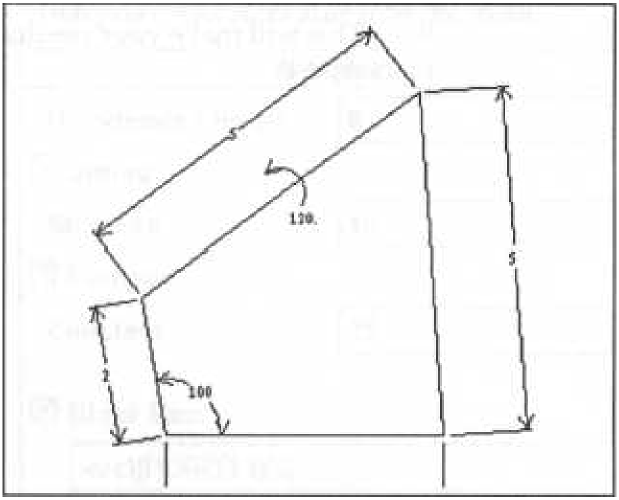
2 - Pick Analysis l Add Load from the menu.

3 - Fill in the value of the torque and its direction (Clockwise or Counterclockwise).



The applied torque symbol is a curved arrow applied to the center of the line.

Torques may be applied to construction lines.



Applied Loads and Fixed Points and Lines

Applied forces and torques are used in performing Static and Dynamic analysis. Analytix will perform either a static equilibrium analysis of the figure or a dynamic equilibrium analysis (if there are masses defined and there is movement).

There is no constraint on the user to enter a balanced set of external forces and torques. Hence Analytix adds balancing external forces and torques so that the entire drawing has zero total force.

Where does Analytix apply these extra forces?

A balancing force is applied to the fixed point and a balancing torque is applied to the fixed line.

This means that unless the external forces and torques which you add have a zero sum, you should make sure you assign the fixed point and line appropriately for your statics or dynamics problem.

If you do not, then Analytix will apply balancing forces and torques to an arbitrary fixed point and line. This will make your results somewhat difficult to interpret.

Analysis / Add Actuator

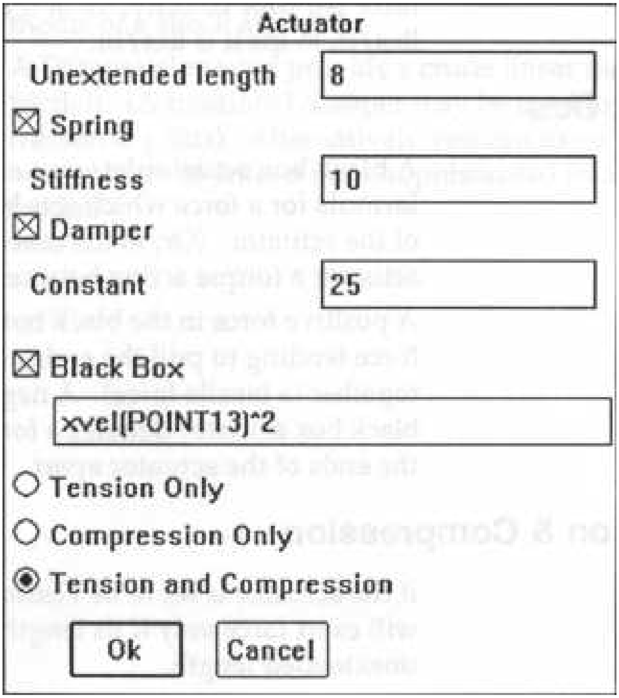
An actuator may be composed of any combination of Spring, Damper, Actuator. It may be translational or rotational.

To create a translational actuator:

Select its two endpoints (holding down the Shift key while selecting the second) then choose Analysis / Add Actuator from the menu.

To create a rotational actuator:

Select the two lines to which it is to be attached (holding down the Shift key while selecting the second) then choose Analysis /Add Actuator from the menu.



When you select Analysis / Add Actuator, you will see the Actuator Dialog Box. This lets you select any combination of spring, damper and black box actuator.

Spring

If you select Spring, you should enter both the unextended length (free length) of the spring and the spring stiffness (spring rate). This stiffness should be in force / distance units. In SI units, this would be N/m, in fps lb/ft, in ips lb/in.

Damper

If you select Damper, you need to enter the damper constant (damping coefficient). This is in units of force per unit velocity. In SI it is in Ns/m, in fps it is lb.s/ft, in ips it is lb.s/in.

Black Box

A black box actuator lets you enter your own formula for a force which acts between the endpoints of the actuator. (Or, in the case of a rotational actuator a torque acting between two lines.)

A positive force in the black box actuator denotes a force tending to pull the ends of the actuator together (a tensile force). A negative force in the black box actuator denotes a force tending to push the ends of the actuator apart.

Tension & Compression

If the actuator is set to be *Tension Only*, the spring will exert force only if its length is greater than its unextended length.

If the actuator is set to be Compression Only, it will exert force only if its length is less than its unextended length.

If the actuator is set to be Tension and Compression, then it will exert a force whether its length is greater or less than its unextended length.

Uses

An actuator may be any one of the types Spring, Damper, Black Box, or it may be any combination:

Spring & Damper

Spring & Black Box

Damper & Black Box

Spring & Damper & Black Box

The Spring & Damper combination provides a useful model of a shock absorber.

A Damper alone can provide a crude linear model of friction. (A rotational damper may be used to model friction at joints). Alternatively you can use a black box actuator to enter a more sophisticated friction model.

Analysis / Resultant Force/Torque

The Analysis / Resultant Force/Torque and Analysis / Force on Pin options are the ways in which you ask Analytix for the results of static or dynamic analysis.

The Analysis / Resultant Force/Torque option is grayed out unless a dimension or actuator is currently selected.

To use Analysis / Resultant Force/Torque:

Select a dimension or actuator.

Pick Analysis / Resultant Force/Torque from the menu.

The Resultant Force/Torque dialog box will appear, which tells you (for a linear dimension or actuator) the force in the dimension / actuator, or (for an angle or angular actuator) the torque in the dimension / actuator.

The force is either a Tension or Compression.

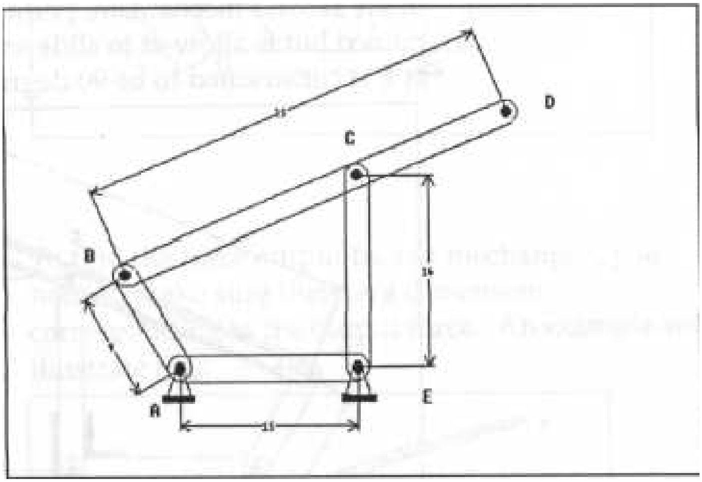
The model where internal forces are transmitted in dimensions may appear a little strange at first, but it is very easy to get used to and you do not need to learn a whole new set of tools to specify how a frame or mechanism is connected. You do this by the way you specify dimensions.

For example, if two bars come together at a welded join which is capable of transmitting moment, then it is appropriate to enter the angle dimension between the lines. The moment transmitted by the join is just the resultant torque in the angle dimension.

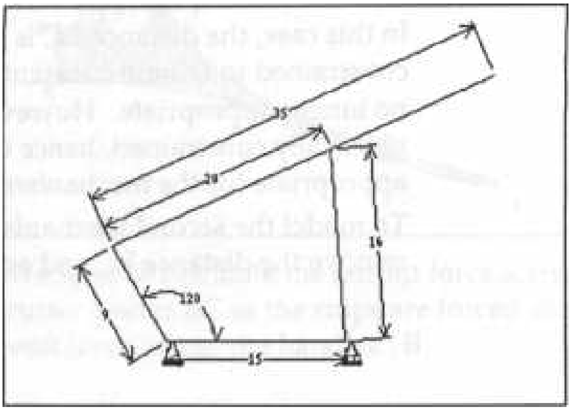
How to dimension for statics & dynamics

We show two different mechanisms, which have the same geometrical representation. The dimensioning scheme reflects the functional difference in the mechanisms.

The first mechanism is a four bar linkage with pinned joints at A,B,C,E. Distance BC is 20. AB is the driving crank.



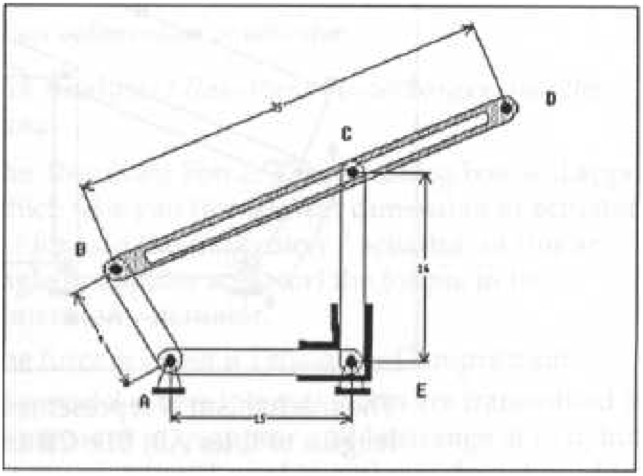
The mechanism is represented by dimensioning lengths of lines AB, BD, CE and AE, and the distance



between B and C. The driving angle BAE is also added.

The rule here is that all the dimensions represent a physical component of the structure which is constraining the structure: the lengths represent the fact that the holes in the bars are set at a fixed distance apart. The angle dimension represents the driving motor which is physically constraining the size of the angle.

In the second mechanism, point C is no longer pinned but is allowed to slide on line BD. the angle at E is constrained to be 90 degrees.



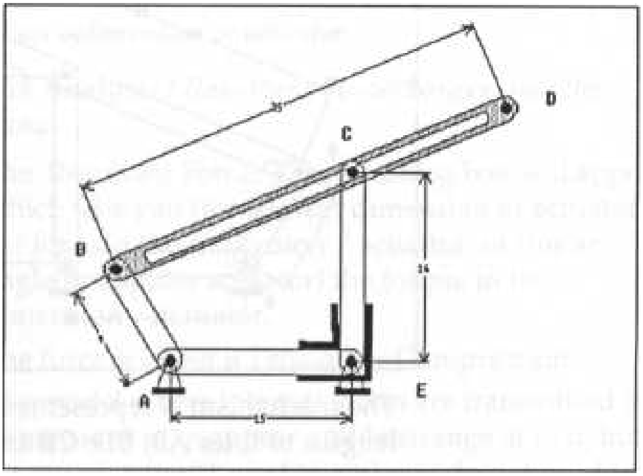
In this case, the distance BC is no longer physically constrained to remain constant, so this dimension is no longer appropriate. However the angle at E is physically constrained, hence this dimension is appropriate for the mechanism.

To model the second mechanism, therefore, we remove the distance BC and add the angle CEA.

between B and C. The driving angle BAE is also added.

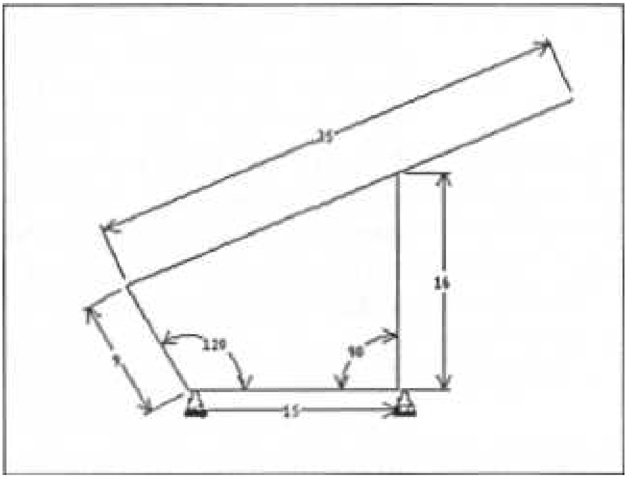
The rule here is that all the dimensions represent a physical component of the structure which is constraining the structure: the lengths represent the fact that the holes in the bars are set at a fixed distance apart. The angle dimension represents the driving motor which is physically constraining the size of the angle.

In the second mechanism, point C is no longer pinned but is allowed to slide on line BD. the angle at E is constrained to be 90 degrees.



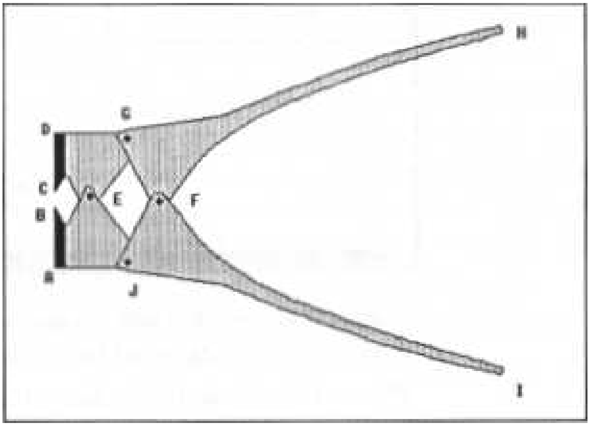
In this case, the distance BC is no longer physically constrained to remain constant, so this dimension is no longer appropriate. However the angle at E is physically constrained, hence this dimension is appropriate for the mechanism.

To model the second mechanism, therefore, we remove the distance BC and add the angle CEA.



Output Forces

To find the force output from a mechanism, you need to make sure there is a dimension corresponding to the output force. An example will illustrate this:



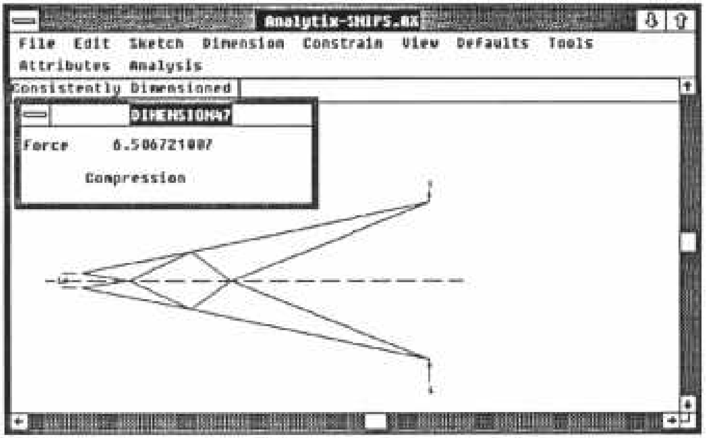
We wish to calculate the output force across the cutter blades BC as the snips are forced shut by a unit force across the handles HI.

We can model the snip mechanism by a set of four triangles, and dimension the triangles by their line lengths. All these dimensions remain constant through the closing of the snips.

We need to add one more dimension to specify how closed the snips are. There are a number of possibilities: we could specify the distance HI between the end of the handles, or the angle HFI, or a number of other angles.

However, as we wish to measure the output force between the ends of the cutter blades, B and C, we should enter this dimension.

In the model, all dimensions except the distance between the cutters have been blanked out. Unit forces have been applied at H and I.



Notice that we did not create any fixed point, which was OK as the applied forces add up to zero.

We could alternatively have made I a fixed point and applied only a force to H. Analytix would have automatically balanced this force with an equal and opposite force applied at the fixed point I. Note that the Calculator function react(dimension) returns the same value as Analysis / Resultant Force/Torque.

Analysis / Stress

Analytix 2.0 allows you to do Stress analysis and Deflection analysis on trusses. The trusses may be simple, compound or redundant. Each member of the truss, however must be a two force member. That is: the lines forming the truss must only be joined at their ends. This means that forces are all along the axis of truss members and there is no bending.

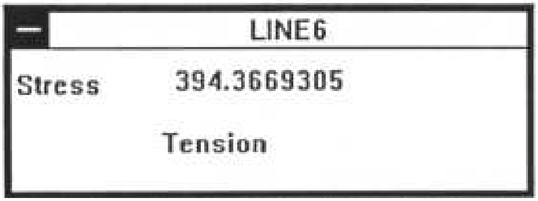
In order to do stress analysis you need first to define a cross sectional area and a modulus of elasticity for each bar in the truss. You can do this for each line individually using Attributes / Info, or you can set them all at once using Defaults / Default Bar Properties.

In contrast to the standard Analytix static model, the truss may be dimensioned in any way. In truss analysis it is the lines rather than the dimensions which are responsible for bearing the loads.

Loads may be applied to the ends of any of the lines representing truss members. Loads may not be applied to points in the middle of lines representing truss members as such loads would conflict with the requirement that the members carry only axial load.

To perform stress analysis, select the line representing the appropriate truss member, then choose Analysis / Stress.

You will see the Stress dialog box.



If you have not specified cross sectional areas and modulus of elasticity for sufficient lines to define a rigid structure, then an error message will appear.

Stress may also be derived using the function:

stress(line1)

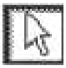
Analysis / Force on a Pin

Use this option to derive the force transmitted through a point of the model.

The total force at a point in equilibrium is of course zero (unless the point has some external forces applied to it.)

What we need to ask for is the force applied to a given point regarded as part of some sub-component by the other components in the drawing.

To use the Force on a Pin option



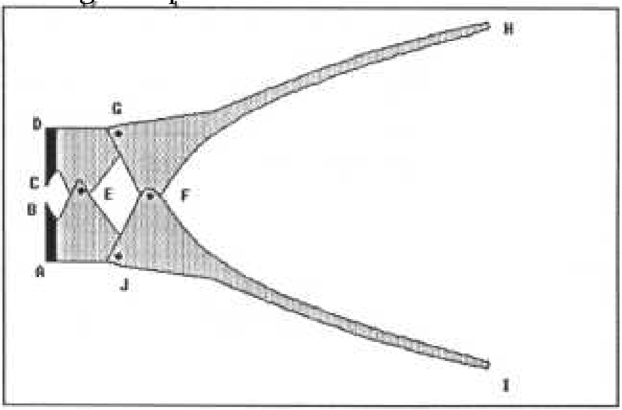
1 - Select the geometric entities which make up the sub-component whose forces you wish to examine.

2 - Pick Analysis / Force on a Pin.

3 - Click on the particular point whose force you wish to examine.

Example

We use the wire cutters described in the above section. We wish to find the force transmitted through the pin at F.

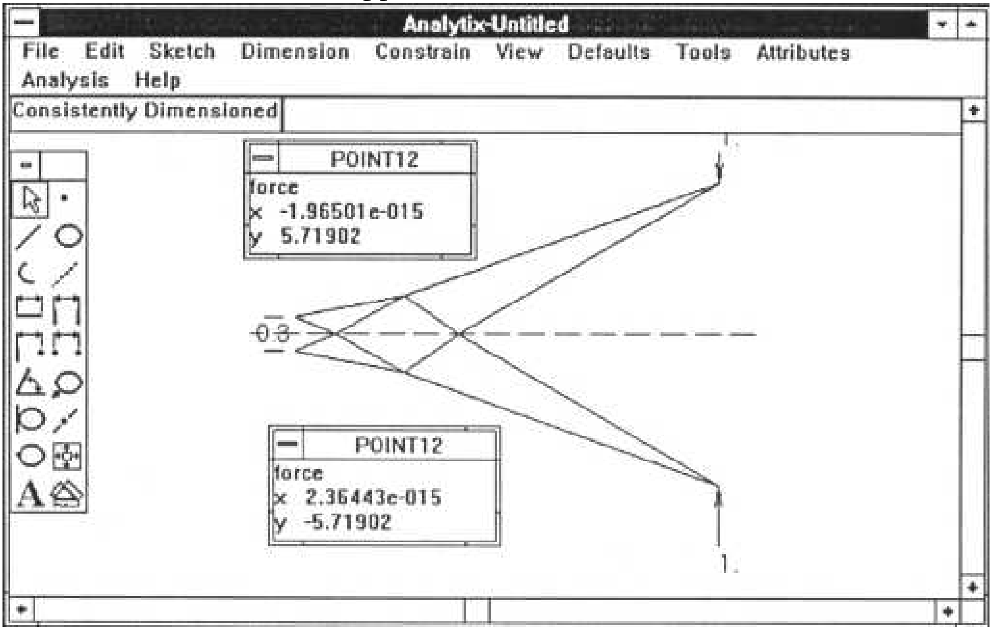


The force at F is transmitted between triangles FGH and FIJ. These forces will be equal and opposite.

We first select triangle FGH, then pick the Analysis l Force on a Pin menu option.

The force box gives us the force exerted by the rest of the diagram (i.e. triangle FIJ) on triangle FGH at F.

If we now select triangle FIJ and ask for the force on the same point, we see that the result is indeed equal and opposite.



More examples of the use of the force analysis capabilities of Analytix can be found in the Examples section of the manual.

Analysis / New Simulation

Simulations are created in two different ways in Analytix.

The Dynamical Analysis module *Dynamix* automatically creates a dynamical simulation. This records at each time step the configuration of a model which is responding to applied dynamical forces.

Alternatively, you can create your own playback simulation using Analysis / New Simulation.

A playback simulation lets you collect a sequence of drawing configurations, then play them back as an animation, or use the simulation in a graph or table.

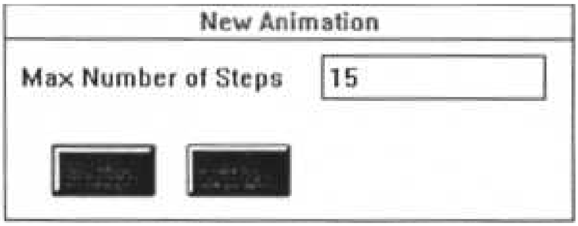
It gives you more flexibility than the regular Animate, Graph and Table tools because it lets you vary more than one dimension. It also lets you vary the dimensions by irregular amounts.

To create a playback simulation:

Select the dimension or dimensions which will vary in the course of the simulation.

Select Analysis / New Simulation.

You will see the New Simulation dialog box, which lets you set the maximum number of steps which may be saved in the simulation.



When you press Ok, you will see the Edit Simulation Dialog Box (See the next section of the manual).

To create a simulation, you first change the dimensions to match the first position of your simulation then add this configuration to the simulation.

You then go back and change the dimensions to match the second position in your simulation then add this configuration to the simulation.

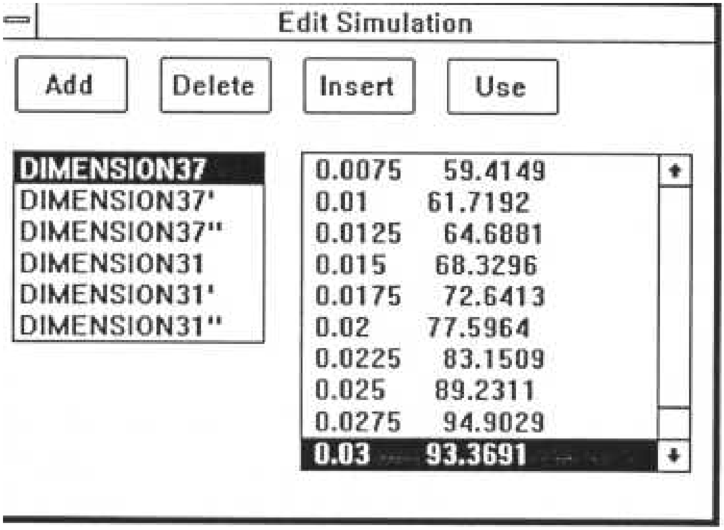
You repeat this procedure till the whole simulation is built.

Note that not only the dimensions themselves but also their velocity and acceleration are stored in the simulation.

Analysis / Edit Simulation

The Analysis / Edit Simulation option lets you create your own playback simulation, or inspect or edit a playback simulation created by the Dynamix add-in.

The Edit Simulation dialog box contains two list boxes. The left hand list box contains the names of the dimensions which vary in the simulation and their derivatives.



In the case above DIMENSION37 and DIMENSION31 vary. DIMENSION37' denotes the first derivative (or velocity) of DIMENSION37. DIMENSION37" denotes the second derivative (or acceleration) of DIMENSION37.

The right hand list box contains the values for the quantity highlighted in the left hand list box throughout the simulation. You use this list box to select a time step of the simulation. You may then Insert a step before it, Delete that time step or Use that time step in your drawing.

In this case DIMENSION37 is highlighted in the left hand list box so the right hand box contains values of this dimension. If DIMENSION37' were highlighted in the left hand box, the right hand box would contain values of the velocity of DIMENSION37.

Add

The Add button allows you to add a step to the end of the current simulation. When you press Add, Analytix will record from the drawing the current values of all the dimensions in the left hand list box and their derivatives. It will add this configuration to the end of the simulation.

Delete

The Delete button will remove from the simulation the time step highlighted in the right hand dialog box.

For example, to remove time step 3:

Highlight time 3 in the right hand dialog box (it does not matter which dimension is displayed). Then press the Delete button.

Insert

The Insert button allows you to insert a step before the currently selected time step in the simulation. When you press Insert, Analytix will record from the drawing the current values of all the dimensions in the left hand list box and their derivatives. It will insert this configuration into the simulation before the time step currently selected in the right hand list box.

Use

When you press the Use button, Analytix will take the dimension values, velocities and accelerations from the simulation for that time step and substitute them into the drawing.

This option lets you go to any specific time step in the simulation and interactively make measurements from the drawing.

Analysis / Tolerance Zone

This analysis option draws the region within which the selected point must lie given the existing tolerances on the drawing's dimensions.

Tolerances on distances and angles are displayed using the Attributes / Measure function.

To create a tolerance zone:

1 - Enter tolerances for the dimensions in the drawing as appropriate.

2 - Make sure that an appropriate point and line are fixed.

3 - Select the point whose tolerance zone you wish to display

4 - Pick Analysis/ Tolerance Zone.

Tolerance analysis can be quite time consuming; so there may be a reasonable lag, then the tolerance zone will be drawn.

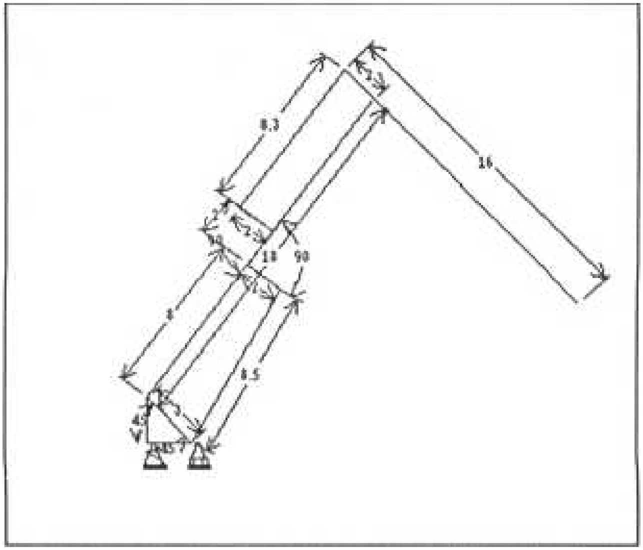
If the tolerances are realistically tight, this zone will probably be too small to see with the naked eye and you will have to zoom in to find it.

A technique which is sometimes useful is to scale the size of the tolerances up by an order of magnitude or two, then you will be able to see both the overall drawing and a scaled up version of the tolerance zone.

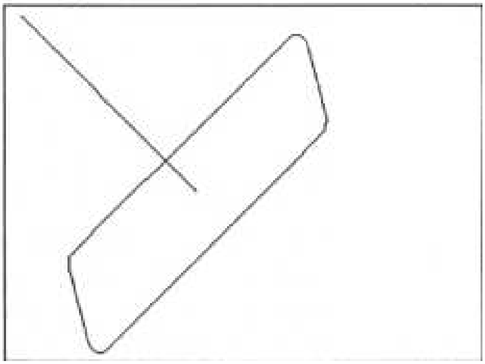
It is important to make sure you have fixed a point and line when performing tolerance zone analysis, as there has to be a fixed reference with respect to which to measure the variation of the selected point.

Max / min tolerance zone

If the current tolerance mode is Max/Min, then the tolerance zone is a polygon. This polygon is the locus of all the positions attainable by the point for within-tolerance values of the dimension.



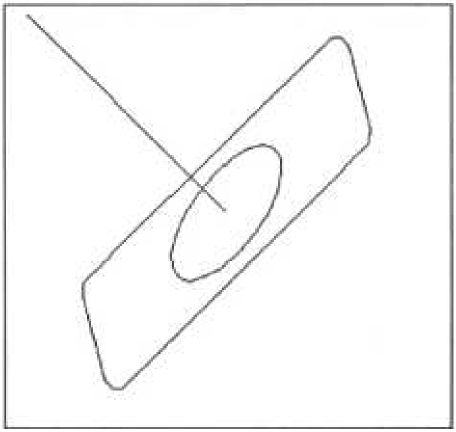
For example, we draw the tolerance zone for the end of the grabber arm of the log loader shown above. All lengths are given tolerances of 0.01, and all angles are given tolerances of 0.25 degrees.



The tolerance zone is a long thin polygon about the point.

Statistical Tolerance Zone

If, in the above example, we switch to Statistical Tolerancing (using the Defaults menu), and draw the tolerance zone for the same point, we see that it is an ellipse nestled inside the polygon representing the absolute tolerance zone.



There are several different ways of describing this ellipse. The most succinct is its mathematical definition: it is an equiprobable ellipse of the Bivariate Normal Distribution which describes, statistically, the position of the point.

This gives you the shape of the region where the point will lie "most of the time". The definition of what is meant by "most of the time" depends on the statistical interpretation of the tolerances which were given for the dimensions.

When you use the Statistical Tolerancing mode, Analytix interprets the upper and lower tolerances as being fixed percentiles of a Normal Distribution. For example you might regard the input tolerances to be 99.9% percentiles - i.e. failure to meet tolerance on a given dimension will occur only 0.1 % of the time. In this case the ellipse shows a region inside which the point will lie 99.9% of the time.

If you interpret the input tolerances to be 95% percentiles, then the output tolerance zone must be interpreted as the region inside which the point will lie 95% of the time.

Erasing Tolerance Zones

Tolerance zones are drawn on the Background level. You erase them using the Edit / Erase Background menu option.

Analysis / Tolerance Trace

This analysis option draws the region of tolerance for a selected coupler curve, given the existing tolerances on the drawing's dimensions.

Tolerances on distances and angles are displayed using the Attributes / Measure function.

To create a tolerance trace:

1 - Enter tolerances for the dimensions in the drawing as appropriate.

2 - Make sure that an appropriate point and line are fixed.

3 - Select the point defining the coupler curve whose tolerance trace you wish to display.

4 - Pick Analysis/ Tolerance Trace.

As with the Tools / Trace function (described earlier in this manual), the Iteration Parameter Box appears.

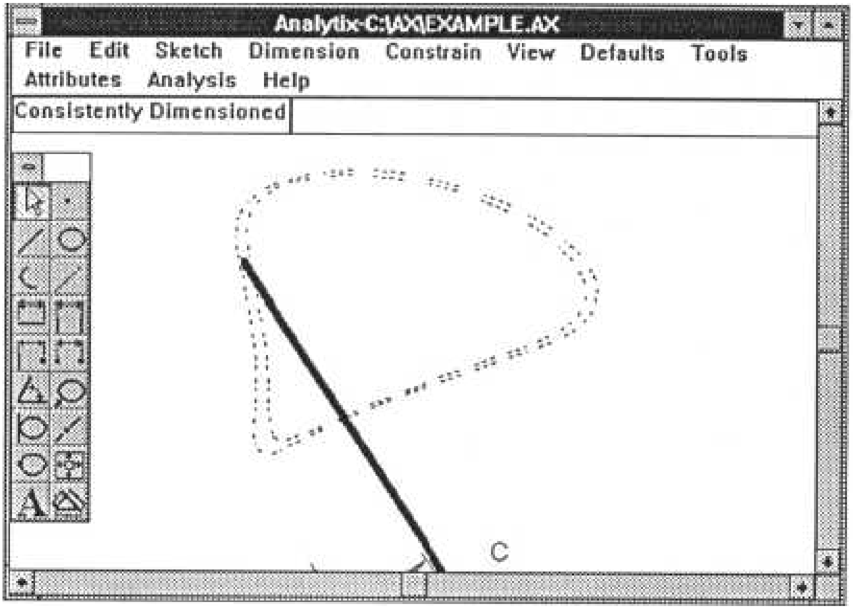
You may use the current simulation if appropriate or fill in the variable or dimension on which the coupler curve depends, its start and end values and step size.

Tolerance analysis can be quite time consuming; so there may be a reasonable lag, then the tolerance trace will be drawn.

If the tolerances are realistically tight, this zone will probably be too small to see with the naked eye and you will have to zoom in to see it in detail.

A technique which is sometimes useful is to scale the size of the tolerances up by an order of magnitude or two, then you will be able to see both the overall drawing and a scaled up version of the tolerance trace.

It is important to make sure you have fixed a point and line when performing tolerance trace analysis, as there has to be a fixed reference with respect to which to measure the variation of the selected point.



Erasing Tolerance Trace

The Tolerance trace is drawn on the Background level. You erase it using the Edit / Erase Background menu option.

Introduction

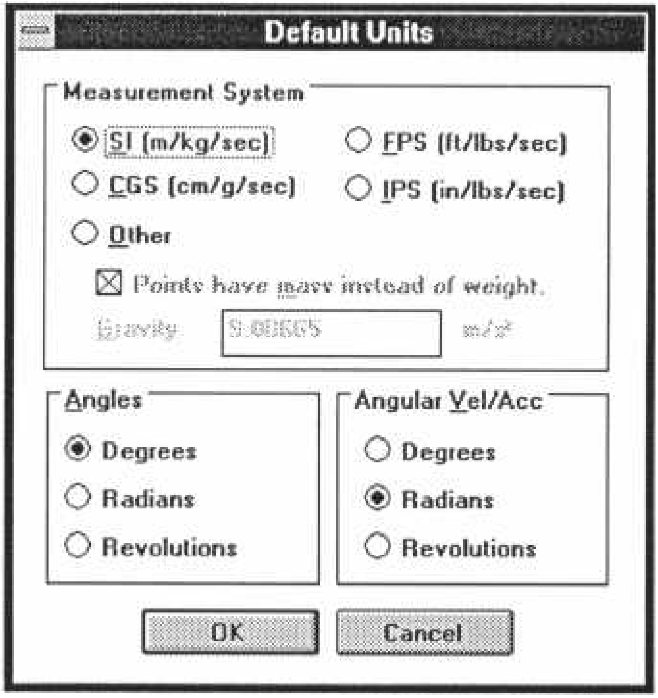
Analytix 3.2 has 3 major additional features, which are documented in this manual addendum:

- Unit Display
- Table Defined Functions
- Cam Support

Unit Display

Analytix 3.2 displays the units, in the current unit system, in the dialog box for any measurement made from the drawing.

You can set the units using the Defaults/Units menu option:



Your options are:

- SI (kg/m/s)
- CGS (cm/g/s)
- FPS (ft/lbs/sec)
- IPS (inch/Ib/sec)

- Other

  If you choose Other, you need to specify the gravitational acceleration, and whether the basic unit is a mass (ass in metric units) or a weight (force) as in English units.

  Here is an example Info Box, with unit information in place:

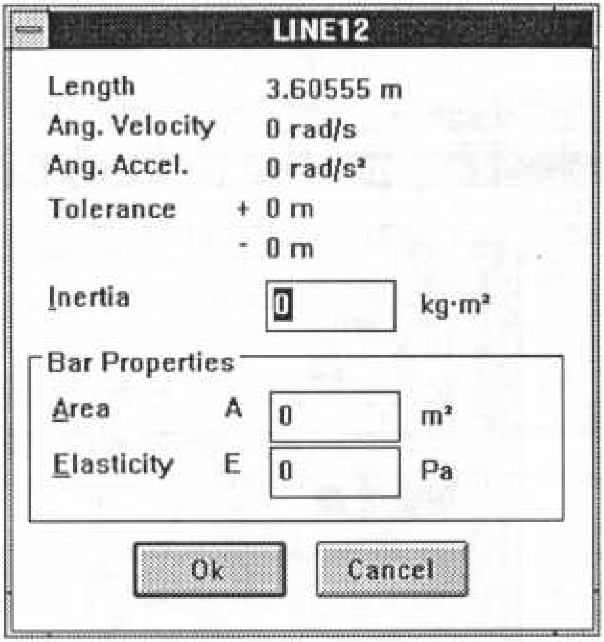
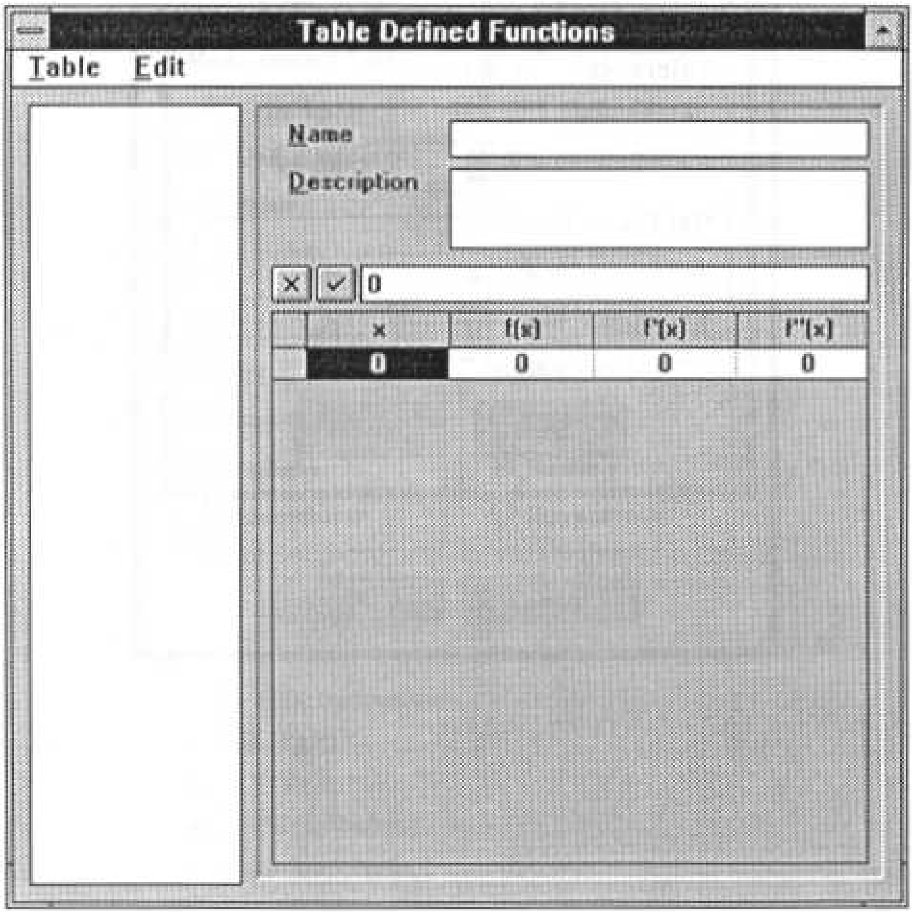


Table Defined Functions

The Table Defined Functions capability allows you to define a function by specifying a set of values. Analytix will then interpolate between these values to evaluate the function for any parameter value.

To invoke the Table Defined Function Dialog, use Tools/Table Functions.

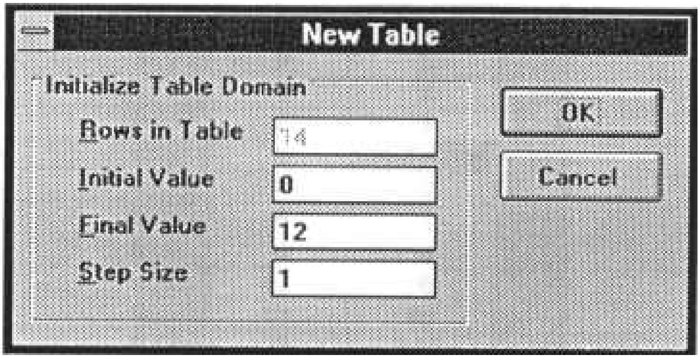


We first describe how to create a simple table defined function. Then we systematically describe the menu options of the Table Defined Functions Window.

How to define a table defined function

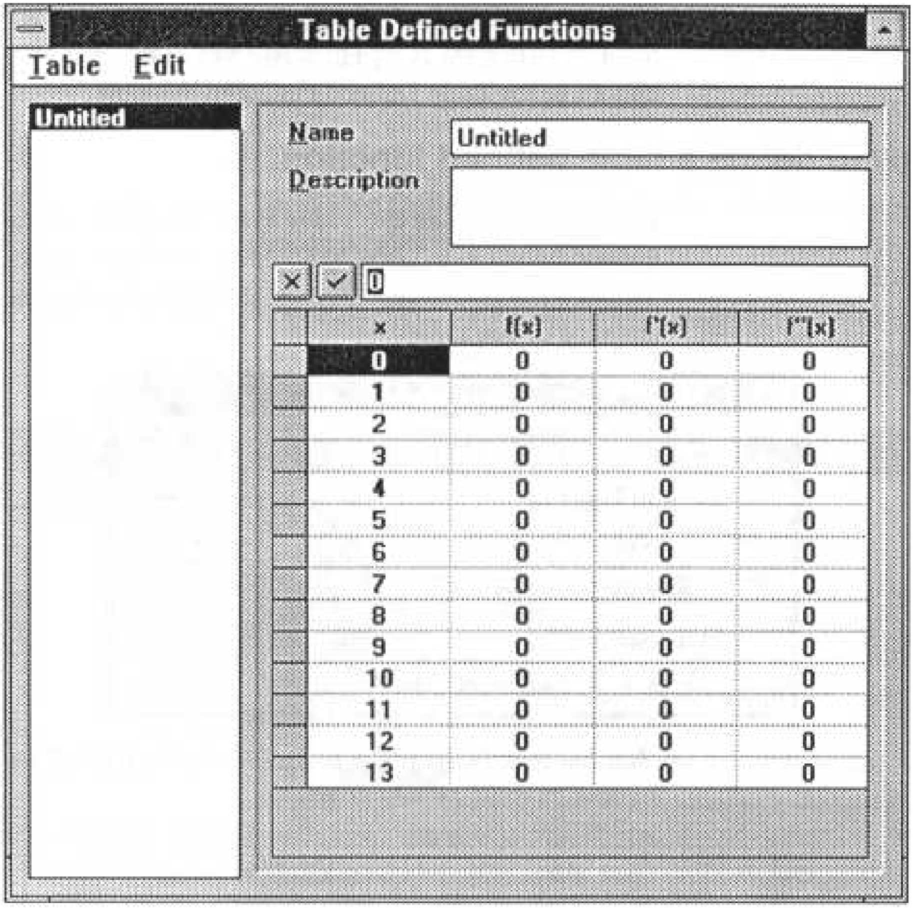
When you first see the Table Functions Window, you should select File/New to create a new function.

You see the following Dialog, which lets you specify the size of the table:



You can specify the initial parameter value and the final parameter value and the step size.

Alternatively, you can specify the number of rows in the table, along with the initial and final values.

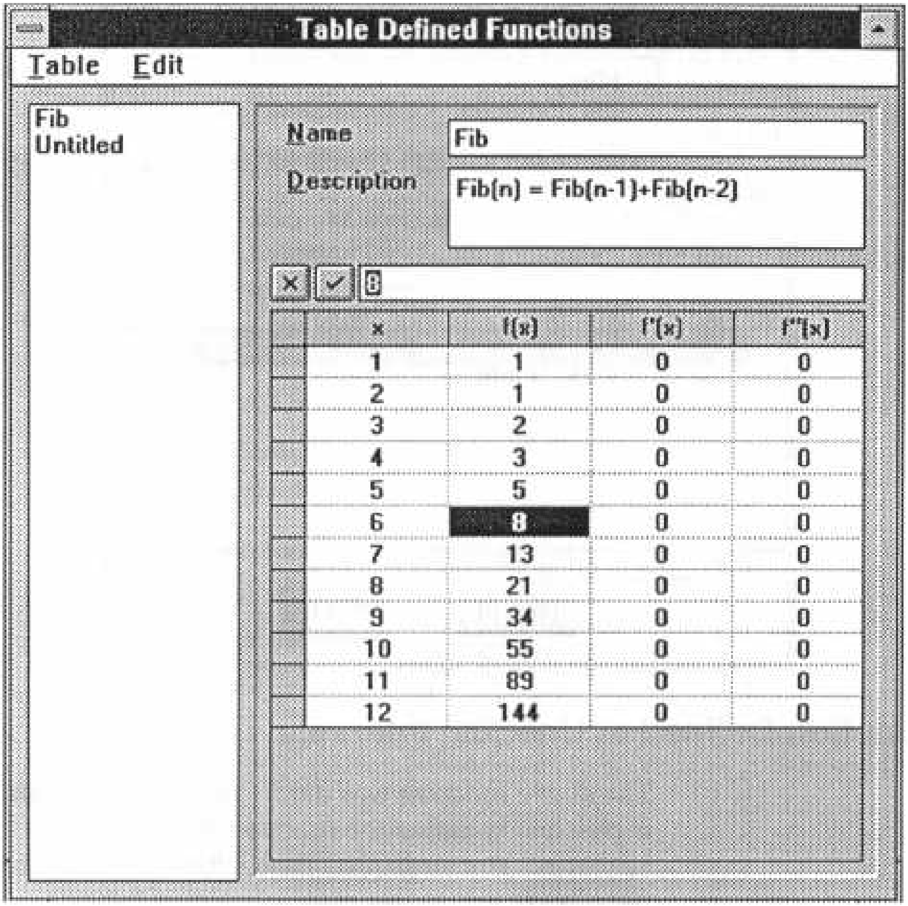


You should give the function a name (the name will be used to invoke the function in calculations).

You may also enter a text description of the function.

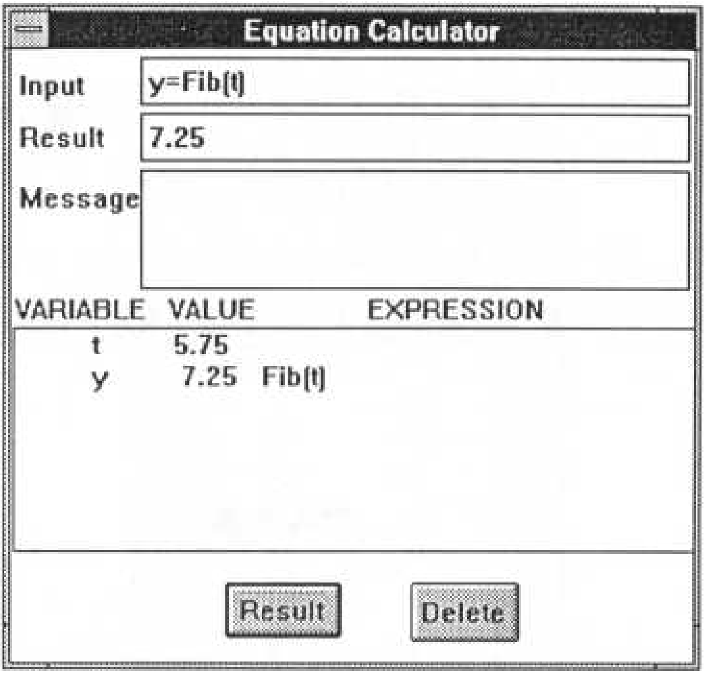
You may then fill in values for the function by clicking on the appropriate place in the grid, and entering the number in the data entry control.

Clicking on the check box accepts the entry. Clicking on the X box rejects the entry.



You can now register this function with Analytix, by selecting File/Update Analytix, or File/Exit to Analytix.

In Analytix, you can now use the function Fib() anywhere you would use a built in function.



Function Defined at Unequal Intervals

The above function was defined at even intervals. It is possible to define the function at irregular intervals. You simply edit the leftmost (x) column of the table.

Derivatives of Table Defined Functions

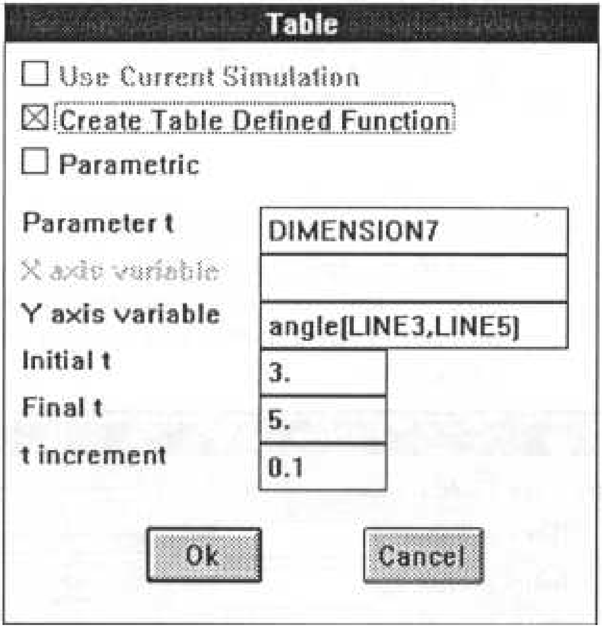
In addition to specifying a function, you can also specify its first and second derivatives. Alternatively you can use Edit/Sort & Differentiate to automatically fill in derivatives computed by finite difference from the table values.

Derivatives are accessed by appending a single quote (') for the first derivative, or a double quote ('') for the second derivative to the function name.

In the above example Fib'(t) would access the first derivative of Fib, and Fib''(t) would access the second derivative.

Creating Functions from Analytix Output

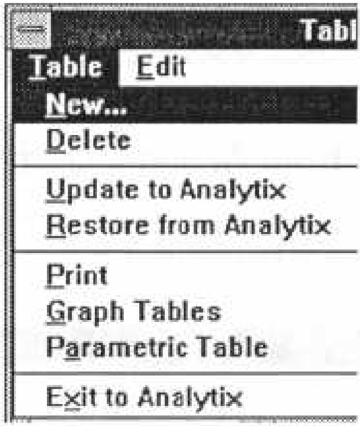
You can turn an Analytix output table into a function by clicking Create Table Defined Function on the Table Dialog.



Note this does not work if the Table Defined Function Window is already open.

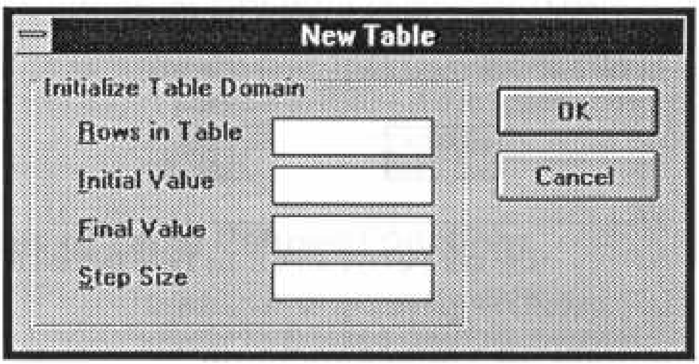
Menu Reference for Table Functions

The following menu options are available for the Table Function window:



Table/New

This menu option lets you create a new table defined function Ax). You are prompted to specify the number of rows in the table and the initial and final values for x



You can fill in any 3 of these values, and the 4th one is automatically provided.

Table/Delete

Use this menu option to delete the current Table Defined Function.

Table/Update To Analytix

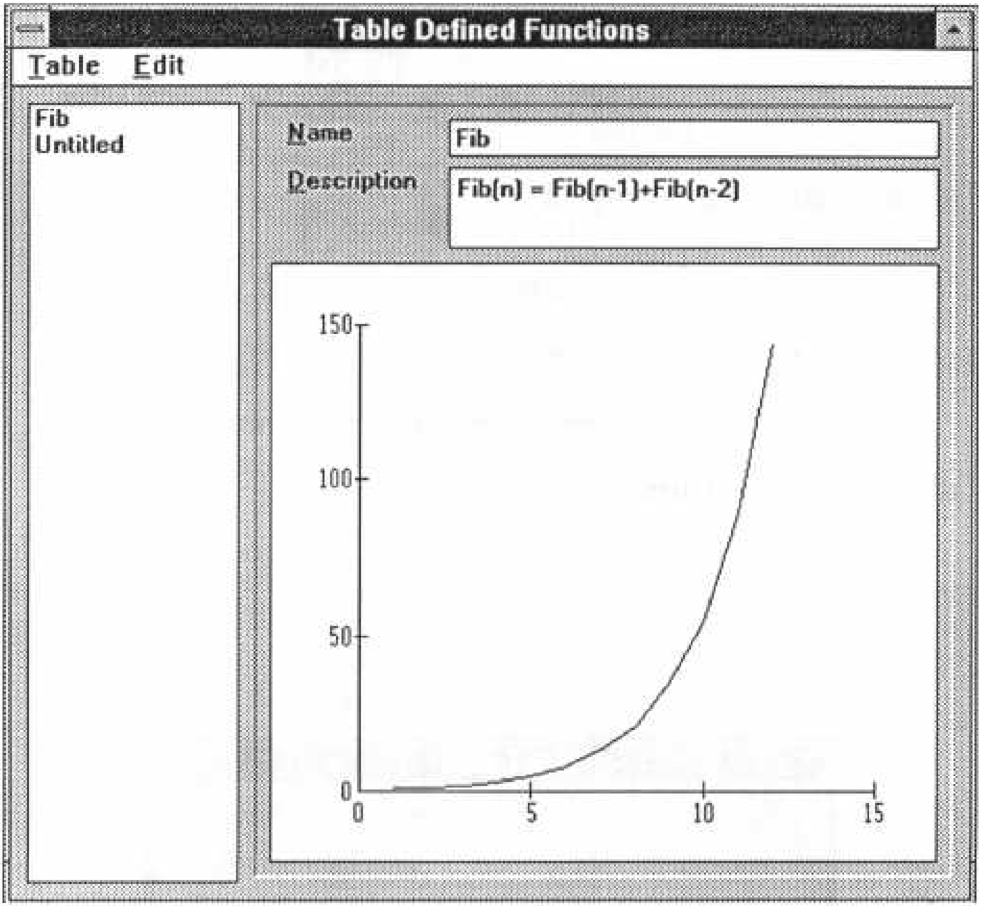
This menu option updates Analytix but does not close the Table Function window.

Table/Print

This menu option prints the current table.

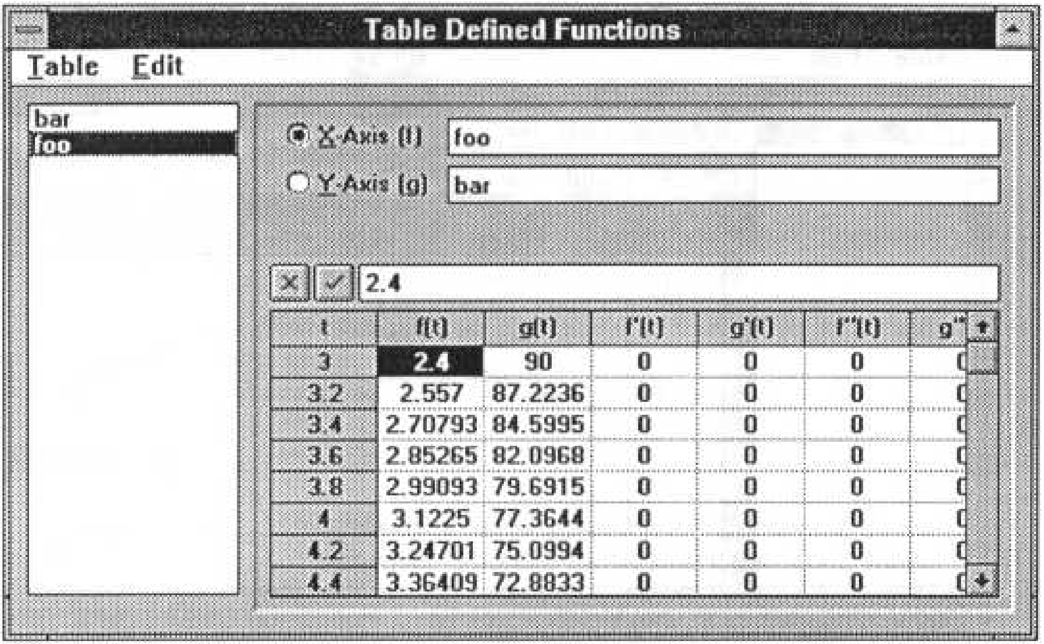
Table/Graph Tables

This menu option graphs the current function.

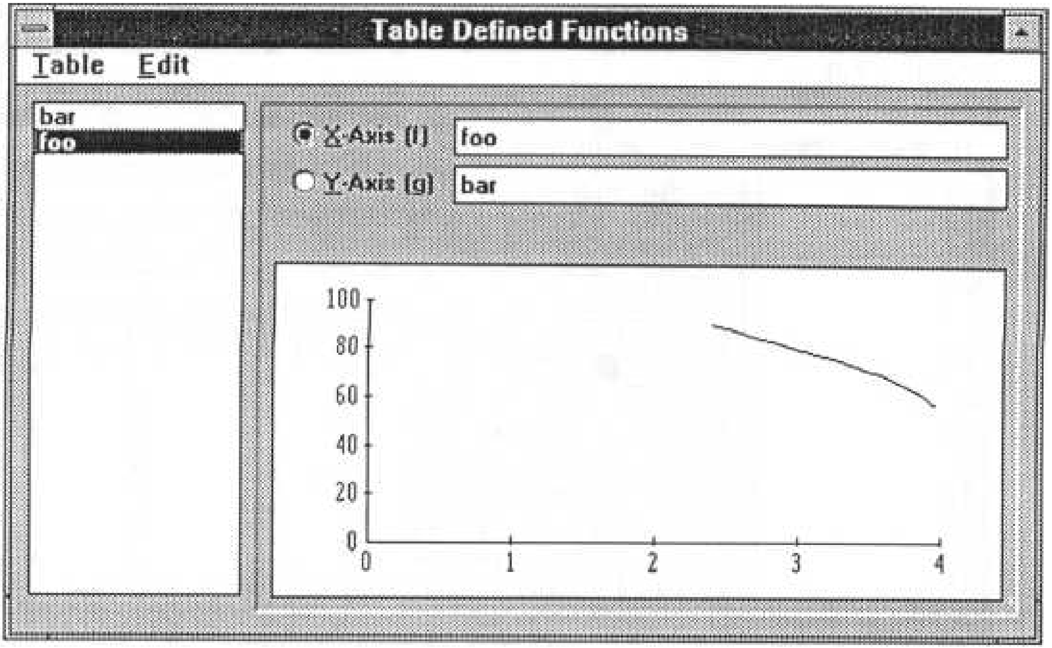


Table/Parametric Table

This option allows you to create 2 table defined functions f(x) and g(x) over the same range of values of x.



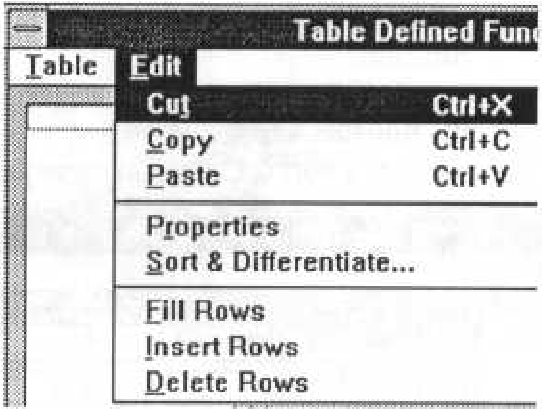
When you invoke the Parametric Table command, a list appears containing all the functions whose domain is the same as the currently selected function. You can select one function for the x axis and one for the y axis of a graph.



The main reason for the parametric table defined function feature is to provide support for invocation of table functions from within a parametric graph or table in Analytix.

Table/Exit to Analytix

Updates the Analytix model and closes the Table Functions window.



Edit/Copy

Copies the currently selected grid items to the Clipboard. You can then Paste these items into another Windows Application, such as a spreadsheet or word processor.

Edit/Paste

Pastes the contents of the Clipboard into the currently selected grid locations.

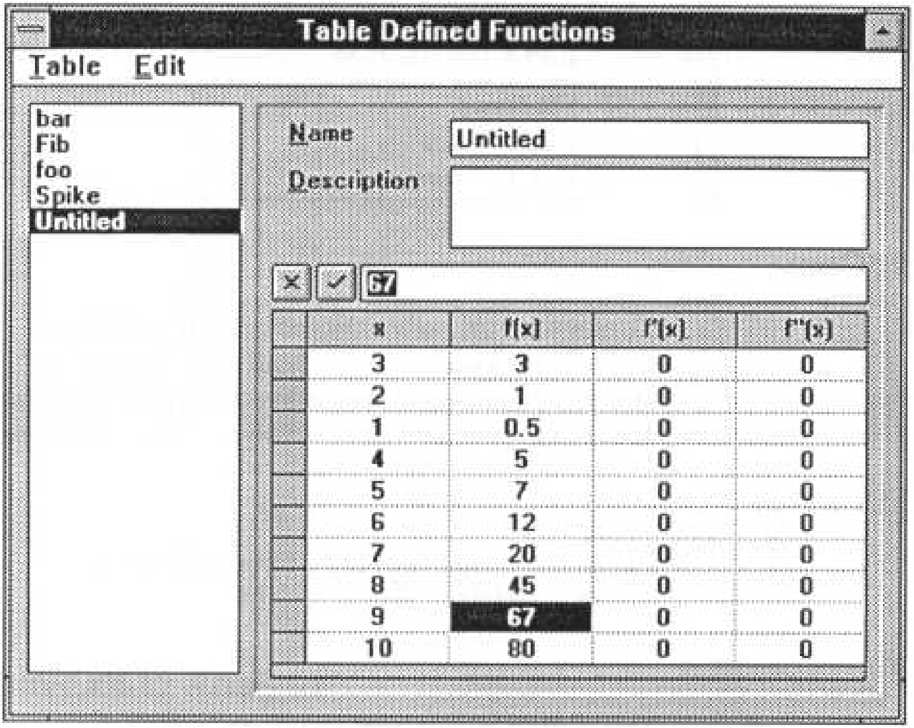
Edit/Properties

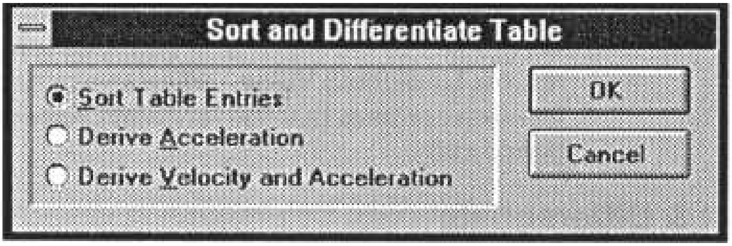
This option lets you select whether your function is bounded or periodic. If you ask for a value outside the domain of the function, a bounded function returns Undefined Value, whereas a periodic function assumes that the function cycles round periodically.

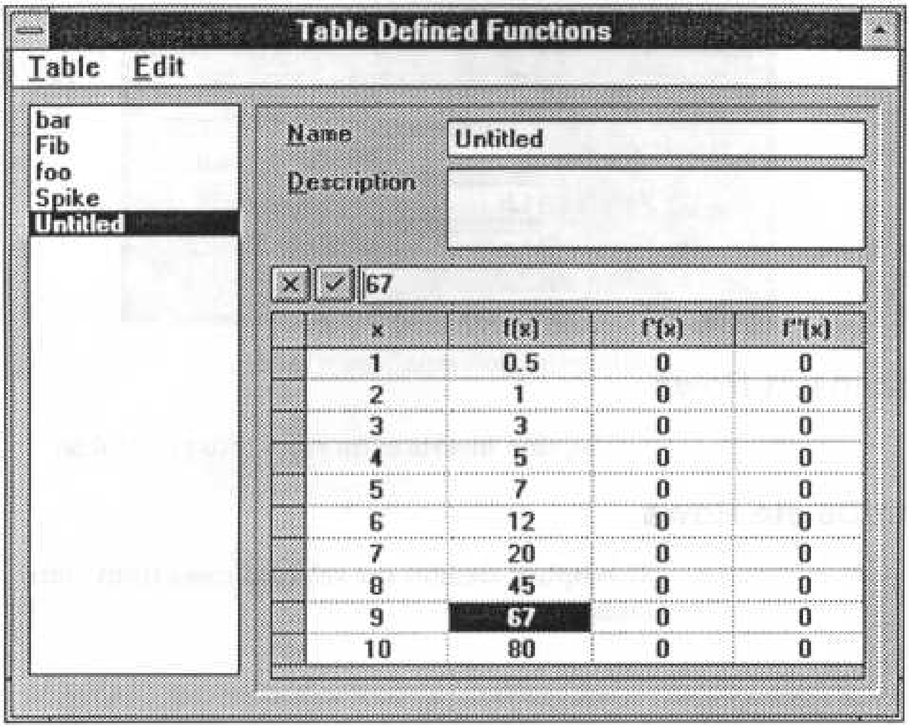
Edit/Sort & Differentiate

This menu option sorts the data points in the domain from smallest to largest. It also fills in values for the first and second derivatives of the function.

In the following example, points in the domain of the function have been entered in a poor order. Applying Sort & Differentiate sequences the points.

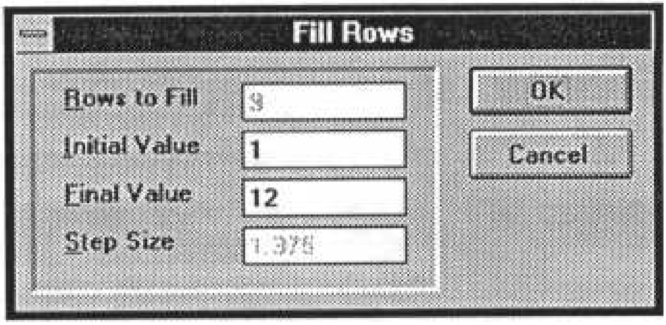






Edit/Fill Rows

This menu option allows you to fill a number of rows with a straight line sequence of values. First select the rows to be filled, then invoke this option. You will be given a dialog box which allows you to specify the values which will be entered:



Edit/Insert Rows

This option inserts extra rows into your table.

Edit/Delete Rows

This option deletes the selected rows from your table.

Cam Support

Analytix 3.2 provides full support for Cams. In order for this option to be available, you need to have the Analytix/Cams package.

The Cam capability is fully documented in the Analytix/Cams documentation.