

PIPE NETWORKS PROGRAM DESCRIPTION

by
Bruce Hunt

The computer program **NETWORKS** can be opened by double clicking on the icon that shows water running out of a tap. This will put the following form on your screen:

Program Pipe Network v2.0 by Dr Bruce Hunt

Number of pipes: 1
Number of nodes: 1
Number of nodes with specified head: 1
Allowable error: 0.0001
Kinematic viscosity:
Acceleration of gravity:
Run Name:

number	head or flow rate
1	
2	
3	
4	
5	
6	
7	
8	

Number	Length	Diameter	roughness height	node number pipe end	node number other pipe end
1					
2					
3					
4					
5					
6					
7					
8					

Edit Box:

Exit Use consistent units Print run

Enter the number of pipes and the total number of nodes in the network as well as the number of nodes at which the piezometric head is specified. (In this case, one node.) The allowable error is used as a criterion for stopping the iterative solution process, and the value of 0.0001 that has already been inserted in the program will probably be satisfactory. (You can test this by rerunning the program a second time with a smaller number, say 0.00001, to see if your answers are significantly different.) All values entered for the kinematic viscosity, the acceleration of gravity, specified heads and specified outflows at nodes must use a consistent set of units. Furthermore, **the program will not accept the use of scientific notation**, so all values except obvious integers, such as the numbers of pipes and nodes, must be entered as real numbers with decimal points.

Under **Node Data** enter either a specified head or a specified outflow for each node. (It is mathematically impossible to specify both the head and outflow at the same node.)

Nodes of specified head must appear first in this list, followed by nodes of specified outflow. Furthermore, outflows must be entered as positive numbers, and inflows must be entered as negative numbers.

Pipe Data entries for each pipe number require the length, diameter, roughness height and node numbers at each end of the pipe under consideration. The order in which these node numbers are entered does not matter, and the pipe length, diameter and roughness height must all have the same length units.

You should enter your name, and a run number if desired, in the **Run Name** box so that you can identify your results when you pick them up from the printer. Click on **Run** to run the program and **Print** to print your results. Input data is written out first in the results, followed by calculated values for the head at each node and then the flow in each pipe. (Flow direction within a pipe is determined by relative values of head at each pipe end but is not stated in the output results.)

Peter Coursey has written this Visual Basic program by translating a Fortran program written by Bruce Hunt. The original Fortran program and the theory behind it are given at the end of chapter 11 of *Fluid Mechanics for Civil Engineers* by Bruce Hunt, which can also be uplifted from <http://www.civil.canterbury.ac.nz/staff/bhunt.asp>