1992

Mesh-Generating Computer Program for the FESWMS-2DH Surface-Water Flow Model

Kirk G. Thibodeaux
U.S. Geological Survey, Stennis Space Center

Published in Hydraulic Engineering: Saving a Threatened Resource—In Search of Solutions: Proceedings of the Hydraulic Engineering sessions at Water Forum '92.
Baltimore, Maryland, August 2–6, 1992. Published by American Society of Civil Engineers.

Mesh-Generating Computer Program for the FESWMS-2DH Surface-Water Flow Model

Kirk G. Thibodeaux[1], Member ASCE

ABSTRACT

A mesh-generating computer program, GNMESH, has been developed to generate two-dimensional finite element meshes for the U.S. Federal Highway Administration's FESWMS-2DH, a two-dimensional depth averaged finite element flow model. GNMESH uses a modified mapping technique that incorporates piecewise Hermitian and weighted interpolation schemes to lay out the generated mesh within given or calculated flow tubes. Minimum required input data for GNMESH is required when the program is run using its default settings. These input data consists of the X, Y, and Z coordinates of points defining a minimum of three cross sections, the roughness coefficients at the given cross sections, an estimate of the water surface elevation at each of the cross sections, and the stream discharge. Output of the GNMESH program consists in part of two data files that are formatted in the manner required by the DINMOD and FLOMOD modules of FESWMS-2DH.

INTRODUCTION

One of the most time consuming steps in using a modeling program that uses the finite element method is the generation of the finite element mesh that defines the system being modeled. Using the Federal Highway Administration's depth averaged finite element flow model FESWMS-2DH is no exception. As described by Froehlich (1989), the bulk of the required input data for the three modules of FESWMS-2DH -- DINMOD, FLOMOD, and ANOMOD -- are the data defining the finite element mesh. This paper briefly describes a computer program developed, GNMESH, that generates two-dimensional finite element meshes for the FESWMS-2DH program. The output files from GNMESH are

[1]Civil Engineer, U.S. Geological Survey, Building 2101, Stennis Space Center, MS 39529.

formatted in such a manner that they can be used as the input files for the DINMOD and FLOMOD modules of FESWMS-2DH with little or no modification.

The GNMESH program was developed so that a minimum of input data is required to obtain a workable mesh. The program can be run with as little input as the X, Y, and Z coordinates of points defining a minimum of three cross sections, the roughness coefficients at the defined cross sections, an estimated water surface elevation at each defined cross section, and the stream discharge. Additional information defining flow tubes and weighting factors can be added to further define the stream being modeled or to force the generated mesh to be denser in specified locations.

GNMESH

When GNMESH is being used with its default settings, the program goes through five steps. These steps are: 1) locate three flow tubes (left[2] overbank, low-water channel, and right[2] overbank), 2) interpolate new cross sections as required, 3) locate corner nodes, 4) lay out the mesh elements, and 5) generate formatted DINMOD and FLOMOD input/control files. When the option of defining the flow tubes is used, the first step is omitted.

Location of flow tubes

GNMESH locates three flow tubes that represent the left overbank, low-water channel, and right overbank by using a combination of ground-surface elevations and roughness zones. This combination is used so that if there is an overgrown secondary channel that has bed elevations close to that of the low-water channel, the low-water channel's lower roughness coefficients will cause the program to choose it as the true low-water channel and assign a flow tube to it. With the low water channel flow tube defined, anything to the left will be defined as the left overbank flow tube and anything to the right will be defined as the right overbank flow tube. Overbank flow tubes with a width of zero are accepted. When the user defines the flow tubes, a maximum of ten flow tubes are permitted.

Interpolation of cross sections

Interpolating additional cross sections takes place in two steps within GNMESH. In the first step the flow tube boundaries between the given cross sections are defined by using a piecewise Hermitian interpolating scheme, described by Lancaster and Salkauskas (1978). In the second step, the X, Y, and Z coordinates of points along the new cross sections are defined by using a linear interpolation scheme based on position ratios.

A Hermitian interpolating scheme is used to interpolate the flow-tube boundary curves because, by definition, the curve being interpolated and its first derivative will be continuous at the given data points. The piecewise Hermitian scheme is used in this program because only two data points, the end points, are needed for each of the flow-tube boundary segments. The curves interpolated between the two end points are smooth, without the wild oscillations that can occur when more than two data points are given per curve segment. The required input data for the piecewise Hermitian scheme are limited to the coordinates of the two end points of the curve segment and the slope of the curve at the two points. By using shape functions derived from the given slopes, data points are easily obtained along the interpolated curve. Only the coordinates of the two end points of the flow-tube boundary curves are required within GNMESH because the program calculates the slope of each boundary curve at its two end points and determines the X coordinates of the new cross section locations along the boundary curves. The slope of at the end points of each boundary curve is calculated by assuming that the defined cross sections on the two ends of the stream section are perpendicular to the stream and that the flow-tube boundary slopes at the ends are the inverse of the slope of the cross section lines. The curve slope at points on the flow-tube boundaries at interior cross sections are taken to be the average of the slopes of two straight lines that join the interior cross-section boundary point to corresponding boundary points on the cross sections on each side of the interior point.

The X coordinates of the points on the flow-tube boundaries of the new cross sections are determined by linearly interpolating along the boundary line between two given cross sections. The number of additional cross sections to be interpolated between given cross sections is determined from the number of elements requested to span the stream and under the assumption of an element length-to-width or aspect ratio of 5 to 1. With these X coordinates, the Y coordinates are then interpolated using the Hermite functions determined earlier.

The data points which define the new interpolated cross sections between the flow tube boundaries are obtained by using a simplified mapping technique. The steps used in this technique are as follows: At the upstream original cross section, the ratio between the distance of a given data point from the left boundary of the flow tube to the width of the flow tube is determined. A data point on the downstream original cross section with the same ratio between its distance from the left boundary and the flow tube's width is located by interpolating between points if needed. For the new cross section, the X and Y coordinates of the data point at the same fraction of the distance from the left boundary of the flow tube are interpolated from the data points obtained earlier at the flow-tube boundaries. Lastly the ground-surface elevation (Z) and the roughness value are linearly interpolated from the data points on the original cross sections. This process is repeated for all of the data points on both original cross sections in each flow tube.

Node location

The element corner nodes are located by a straightforward method that uses the inverse of the roughness coefficients as a weighting factor when interpolating.

[2]Left and right are defined by the observer as looking downstream.

The nodes are located so that the weighted area of the element face between two corner nodes bounded by the ground surface and the water surface, is equal to the total weighted cross sectional area divided by the number of elements across the stream. Additional weighting factors can be added by the user, if needed, to place more elements into regions where the gradients of dependent variables are expected to be large. Nodes can also be forced to be placed at breaks in the bed slope by defining these points as additional flow tube boundaries.

Element layout

The most desirable element shape for shallow-flow finite elements that are available for use in FESWMS-2DH, as described by Lee and Froehlich (1986), is that of a 9-node quadrangular element because it is claimed to be the most accurate element shape for calculating both velocity and pressures. In addition, Froehlich (1989) points out that because of boundaries and varying element sizes, most finite element networks can and should be a mixture of 6-node triangular and 9-node quadrangular elements. This combination of element types will provide the best representation of the water body being modeled. Because quadrangular elements are the most desirable element shape, GNMESH will lays out quadrangular elements wherever possible and use triangular elements when needed.

The procedure used in GNMESH assigns the four corner nodes to a quadrangular element and fills in only the node numbers for the mid-side and center nodes. The DINMOD module of FESWMS-2DH interpolates the X, Y, and Z coordinate values of the mid-side and corner nodes. When a triangular element is needed, GNMESH assigns the three corner nodes and again lets DINMOD assign the mid-side node coordinates. A simple rule followed by the program is that no element will cross the boundary of a flow tube, thus keeping elements from spanning into different flow condition zones. Triangular elements are used when a flow tube is reduced or enlarged in width between one cross section to the next. Figure 1 in the example below demonstrates how the elements are laid out to form a finite element mesh.

Output files

Three files are output by GNMESH. These are a general output file that holds calculated values of the interpolated cross sections, and two files that are formatted as FESWMS-2DH input files.

The file formatted as input for DINMOD contains the program-control data set, element data set, node data set, and element-resequencing data set. The program-control data set contains default values for the job option codes which control the overall operation of the DINMOD program. The default values were chosen to be applicable to typical conditions encountered in running the program. The element and node data sets contain the information calculated by the GNMESH program. The element-resequencing data set contains a set of default control instructions and four lists of elements to begin ordering the elements. The

resequencing portion of DINMOD is used to determine a more efficient solution when using the frontal solution method in solving the finite element.

The file formatted as input for FLOMOD contains the program-control data set, property data set, total-flow cross-section data set, water-surface-elevation cross-section data set and the flow-check data set. The program-control data set contains default values for the job option codes which control the overall operation of the FLOMOD program. The default values used were chosen on the basis of many runs using a variety of streams. Because the default data set is intended to apply to a variety of conditions, the values are intended to be used only as a starting point. One or two of the values will usually have to be changed for individual sites.

The property data set consists of data records associated with the property codes that are assigned to the mesh elements when the elements are laid out by GNMESH. Each record consists of a property code number, values for Manning's roughness coefficients, the water depth at which the roughness coefficients are applied, a Chézy discharge coefficient, and the turbulence kinematic eddy viscosity values. The Chézy discharge coefficient is not used or calculated by GNMESH and is assigned a value of zero. The turbulence kinematic eddy coefficients assigned by GNMESH are default values for typical model conditions. These turbulence coefficients are intended for a "cold" start and must be changed to values closer to those suggested by Froehlich (1989) after the first few iterations of the model.

The total-flow cross-section data set (QSEC) consists of a list of node numbers that define a connected series of element sides that represent the upstream-boundary cross section. GNMESH generates the node list from the farthest upstream cross section and assigns to the list a cross-section identification number and the total flow that is to be applied across the cross section. GNMESH assumes that the stream discharge is the total flow.

The water-surface-elevation cross-section data set, like the QSEC, consists of a list of node numbers that define a connected series of elements that represent, in this case, the downstream boundary. The cross section used in this data set is the farthest downstream cross section. For this data set GNMESH assumes a constant water-surface elevation across the cross section and outputs the cross-section identification number, water surface elevation, and boundary condition code.

The flow-check data set consists of lists of node numbers that define a line of element sides across which the total flow is calculated. The default output from GNMESH are lists of nodes that define the inflow and outflow boundaries.

EXAMPLE

To demonstrate the type of mesh network output by GNMESH, the data from a stream section with three initial cross sections is used as input to the program. The left overbank of this stream section reduces to a width of zero at the center of the section and then increases in width towards the downstream exit as the low-water channel meanders across the flood plain (Figure 1). The number of

elements requested across the flood plain was 15 with 3 flow tubes specified. The number of elements for this example is 125 elements with 489 nodes. Using the default settings in GNMESH on this stream section produces a mesh consisting of 3863 nodes and 976 elements. The largest mesh generated to date by GNMESH consisted of 19159 nodes and 4822 elements.

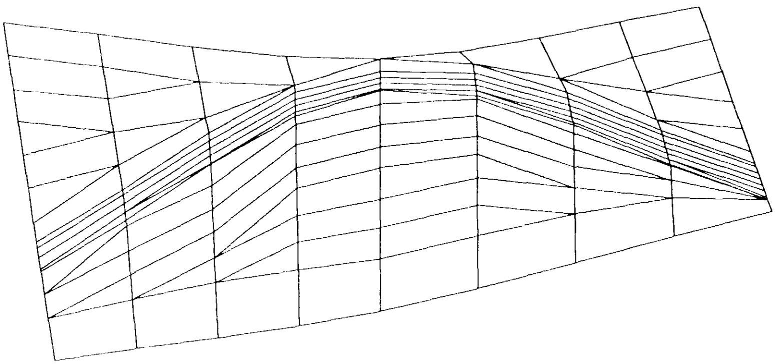

Figure 1. Sample finite element mesh generated by GNMESH.

CONCLUSION

A mesh generating computer program, GNMESH, has been developed to generate a finite element mesh for the U.S. Federal Highway Administration's FESWMS-2DH, a two-dimensional finite element flow model. The GNMESH program is a relatively simple program to use because it requires a minimum of input data. The mesh output from the mesh generator, with little modification, can the be used when running the FESWMS-2DH program.

APPENDIX - REFERENCES

Froehlich, David C. (1989). Finite element surface-water modeling system: Two-dimensional flow in a horizontal plane - Users manual. *Publication No. FHWA-RD-88-177*, Federal Highway Administration.

Lancaster, P. and Salkauskas, K. (1978). *A Survey of curve and surface fitting.* University of Calgary, Canada.

Lee, Jonathan K. and Froehlich, David C. (1986). Review of literature on the finite-element solution of the equations of two-dimensional surface-water flow in the horizontal plane. U.S. Geological Survey Circular 1009.