# AUTOMATIC MESH GENERATION OF JOINT / INTERFACE ELEMENTS

N.C.Thai[1], T.Hasegawa[2], S.Aoyama[1] , and Z.L.Tun[1]

[1] *Division of Environmental Science and Technology, Kyoto University*
*Sakyo-ku, Kyoto, 606 , Japan*
[2] *Department of Natural Resources Management, Kinki University*
*Nakamachi, Nara City, Nara 613-8052, Japan*

**SUMMARY**: It is often difficult and requires a lot of data to generate interface element of zero or very thin thickness. In this paper a method which automatically determine joint/interface element positions based on material properties of elements is presented. By recording elements around nodal points in a special way, additional nodal points and interface positions are correctly determined. Additional data required is very small. The method is applicable to all one-dimensional and two-dimensional element types and can easily be incorporated into available automatic meshing programs with minor modifications. It was used to generate interface elements of several complicated domains such as in composite materials or geosynthetic reinforced soils successfully. The method also can be extended to three-dimensional cases.

**KEYWORDS**: Finite element, joint / interface element, automatic meshing

## INTRODUCTION

The uses of interface elements with zero or very thin thickness in several branches of engineering are increasing, especially in geotechnical engineering to model soil-structure, soil-geosynthetic interactive or composite material in finite element analyses. At the contact surface of two different materials if stiffness of one material is several orders higher than of the other, analysis using standard two-dimensional elements will cause numerical problems. In this case interface or joint elements should be used to allow relative movement to occur. Mesh generation for interface element is very tedious and time consuming, it should be done by an automatic mesh generation program. Despite there are a lot of algorithms for two-dimensional mesh generation, technique for automatic mesh generation of zero or thin thickness interface element has been rarely mentioned in literature. Therefor simple technique, which can be easily incorporated into available automatic meshing programs, is needed.

It has been generally accepted that the best way to generate a mesh with interface elements was to generate a mesh of two-dimensional elements first without considering joint and interface elements then additional nodal points and interface elements are added into the basic mesh. To generate joint or interface elements, normally joint positions have to be specified along certain lines or element block sides [1], [2]. It is a simple task to add nodal points and interface elements along a block side but intersections of interface elements pose particular problems. To overcome the difficulty in dealing with intersection of interfaces Potts et al. [2] proposed an interesting method which involve opening and closing cracks along two dimensional element blocks as initial determination of interface element positions. However it would be very difficult to apply this method to complicated domains such as in composite materials or in geo-synthetic reinforced soil for the two following reasons:

- Firstly, in these cases, so many interfaces exist and their positions are not only located on smooth curves but also on zigzag courses. Many interfaces may intersect at a point. It would require a lot of data to specify interface positions. Moreover in case of composite with inclusions randomly distributed, for example in case of natural formed composite such as geo-materials or concrete materials, there are a lot of inclusion with various sizes and shapes with different volume fractions. Positions of these inclusions are determined by a random process [3], [4]. It is impossible to know interface positions in advance.

- Secondly, the method involve in closing of unused cracks and removing redundant nodal points is not simple in terms of computer program implementation. Thus a technique which requires simple data, easy implementation, and can be applied in general cases would be preferable.

Here in this paper a method that can automatically search interface locations and then generate joint or interface elements based on material properties of each element is presented.

## DATA PREPERATION

It is assumed that a two dimensional auto-meshing program is available. The techniques for automatic mesh generation of two-dimension domain have been discussed by several authors and will not be mentioned here. Using a two-dimensional automatic meshing program, information such as element number, element's nodal points, co-ordinates of nodal points, and element connectivity, etc., are easy to obtain. Fig. 1 shows a two-dimensional mesh, which comprises of two different materials. Material properties of elements 3, 7 and 8 are different with other's. For purpose of interface determination, an array, which stores information about elements surrounding each nodal point, is needed. The order of elements surrounding a nodal point should satisfy two requirements:

- Array should be arranged in counter clockwise direction and elements located continuously. For example elements surrounding nodal points in Fig. 1 should be arranged in the order (2, 3, 1) for node 2, (1, 3, 6) for node 5, and (7, 8, 9, 10, 6, 3, 2, 5) for node 7.
- Array of elements should start from an element such that the left neighboring element has different properties. With the first requirement, this requirement is automatically satisfied for nodes on the boundary. For internal nodes, which are fully surrounded, there may be several possible choices, for example elements around node 7 can be arranged as (3, 2, 5, 7, 8, 9,1 0, 6), (7, 8, 9, 10, 6, 3, 2, 5) or (9, 10, 6, 3, 2, 5, 7, 8), etc.

By arranging elements around nodal point in this way, it is easy to determine number of nodal points being added to the mesh systematically as will be shown in subsequent section. This information is considered as the most important indicator to determine joint / interface

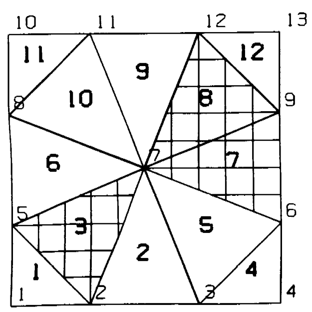


*Fig. 1*

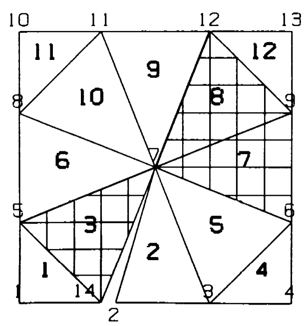


*Fig. 2a*

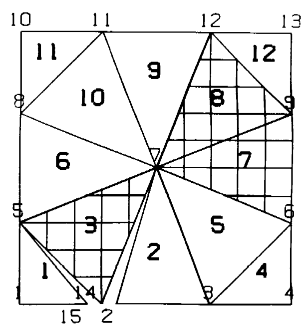


*Fig. 2b*

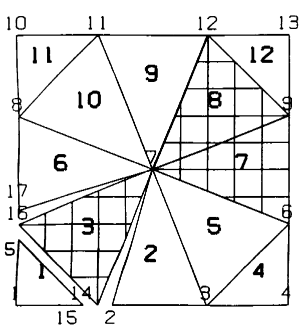


*Fig. 2c*

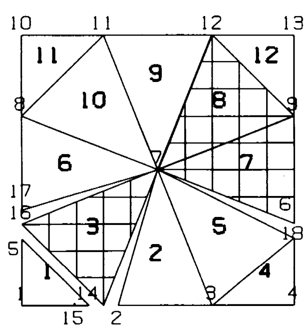


*Fig. 2d*

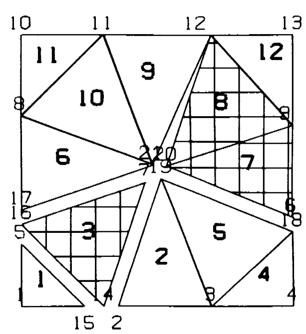


*Fig. 2e*

element positions. It is, however, noted that the above requirements are only necessary for nodes whose surrounding elements have different properties. If all surrounding elements have the same material properties then the order of the array is of no more importance. Each element is assigned a certain material number depending on its position. An artificial material property, for example Young's modulus, is given to each material number. A ratio of Young's modulus between two adjacent elements defined as R = E1 / E2 where E1 is Young's modulus of the first element and E2 is Young's modulus of the second element. When R or 1/R is greater than a certain value $R_{limit}$ decided by user, then an interface is created. The reason for use of an artificial material property is that not all contact surfaces between two different materials need to be modeled by interface elements. As a matter of fact, in practice interface elements are used at interface of two materials with very much difference in property only. The user can control whether interface elements should be used by simply giving a suitable limit ratio $R_{limit}$ of Young’s modulus between different materials. The values of artificial material properties are fictitious and will not be used for analysis. In the analysis, actually material properties based on material number of each element will be used.

Among data required for interface element generation mentioned above, elements around nodal points and element material number are available by any two-dimensional meshing program with very small modifications. Only artificial Young's moduli of materials are considered as additional data. In fact, even in complicated cases number of different materials is generally less than 10, therefore additional data are very small compared to data for defining interface element positions.

## JOINT / INTERFACE ELEMENT GENERATION

In this proposed method joint/interface positions and number of additional nodal points will be determined automatically based on material properties of elements only. The process is divided into two steps; addition of nodal points and insertion of joint/interface elements.

### Addition of nodal points

The first step in the interface element generation process is addition of nodal point to the existing mesh. Each nodal point will be checked to see if there is any pair of elements, which sharing the node, have different Young's module with ratio over limit condition. If such pairs exist, for example nodes 2, 5, 6, 7, 9, 12 in Fig. 1, these nodes will be recorded in an array for future use. At this type of nodes, new nodal points with the same co-ordinates will be added. After all nodal points have been checked, the process of adding nodal points begins. This time only nodal points, which have been marked before, are considered. Each nodal point is considered in turn.

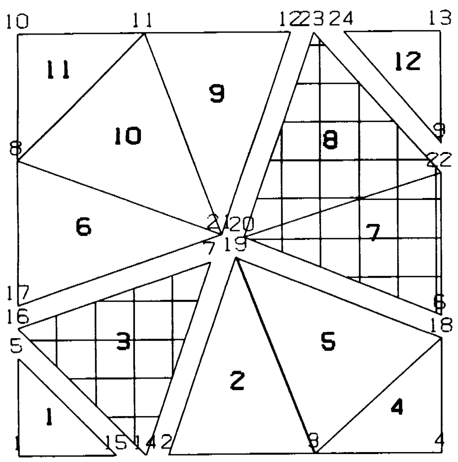

*Fig. 3a*

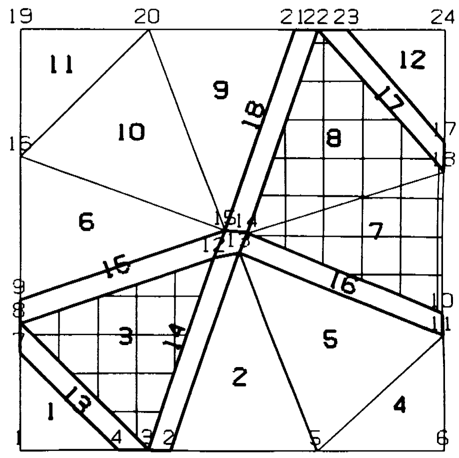

*Fig. 3b*

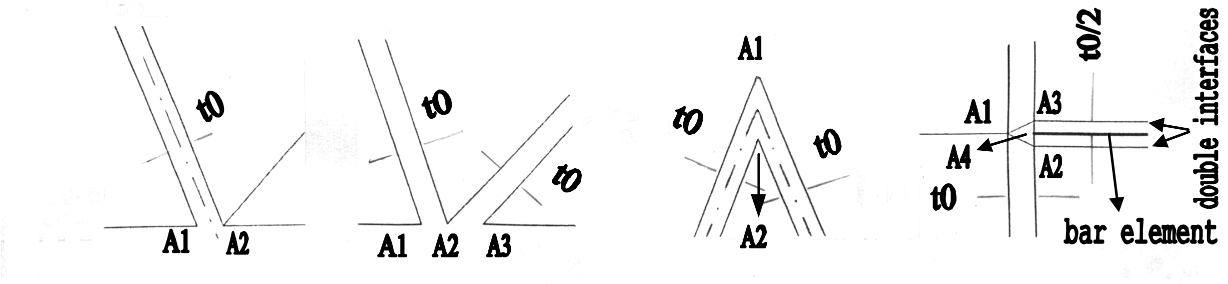


*Fig. 4a* *Fig. 4b* *Fig. 4c* *Fig. 4d*

Now let us consider any node N, starting from the second element in array of elements surrounding node N, the Young's modulus ratio R of current element and element located before it in the array is determined. If $R \geq R_{limit}$ a new nodal point N1 will be added. The new node is numbered consequently with co-ordinates the same as those of node N. All elements from current element to the last element in array will have element connectivity updated to the new node. It means that these elements now have N1 as an element nodal point instead of N. If R is smaller than $R_{limit}$ nothing will change and next element will be checked until the last element in the array. Fig. 2a, 2b show how new nodes are determined and element connectivity is updated. Node 2 in Fig. 1 is surrounded by elements (2, 3, 1) in counter clockwise order. Searching starts from element 3, because element 3 and element 2 have different material properties, a new node numbered 14 is added. Element connectivity of element 3 and 1 are (2, 7, 5) and (1, 2, 5) respectively are updated to (14, 7, 5) and (1, 14, 5). Subsequently element 1 being compared with element 3, this time material properties are also different and once more a new node numbered 15 is added at the same position with node 2. Element connectivity of element 1 is then updated to (1, 15, 5). Actually in Fig. 2a, 2b node 14 and 15 coincide with node 2, but for the sake of clarity they are separated into three different locations.

The process continue with other nodes until all nodal points have been checked as can be seen in Fig. 2a, 2b, 2c, 2d, 2e. Finally all nodal point positions can be seen as in Fig. 3a. The two requirements for arrangement of element around nodal point ensure additional nodal points will be properly inserted

**Insertion of interface elements**

After all necessary additional nodal points are inserted and element connectivity updated accordingly, the next step is to generate interface elements. The only remaining thing is the formation of interface element connectivity correctly. Interface element type i.e. 4-node element or 6-node element depends on two dimensional element types. If the two-dimensional mesh uses 3-node triangular elements and 4-node quadrilateral elements, 4-node joint/interface elements will be used. This time all two-dimensional elements are checked. For each element, its material will be compared with neighboring elements. To avoid interface element being inserted repeatedly the comparisons are carried out with only neighboring elements of larger element number. If any neighboring element have different material properties, along the side shared by two elements an interface element will be inserted. As a result of the first step, these elements now do not share any common node, they are effectively separated, a virtual place exist this is where interface element is located. Element nodal points are counted in counter clockwise direction these nodes belong to above mentioned elements and lie on interface. Since all element connectivities are recorded in counter clockwise direction, it is very easy to determine order of interface element connectivity.

**Renumbering nodal points**:

Because additional nodal points are numbered consequently the difference of order between nodal points of element become large especially with cases of large mesh.

To reduce bandwidth of total stiffness matrix it is necessary to renumber of nodal points. Fig. 3b shows a completed mesh after generation of interface elements and renumbering of nodal points.

## THIN LAYER INTERFACE ELEMENT

Currently, beside use of joint/interface of zero thickness, the use of thin layer interface element is increasing because it has several advantages over zero thickness interface element [5]. To generate finite element mesh of thin layer interface with thickness *t0*, first a finite element mesh of joint / interface with zero thickness is generated then interface elements are expanded by moving interface nodal points to new positions. Nodal points, which are not belonging to any interface element, are not moved. To avoid distortion of mesh, some rules for shifting interface nodal points are adopted as follows:

- For nodes at a boundary, if at these node positions only one joint / interface exist (single interface) (point A1, A2 in Fig. 4.a) nodal points will move along boundary and expand the interface by a half of the thickness. If at nodal point position two interface elements intersect, the node common to both interface elements is not shifted (node A2 in Fig. 4b) other nodes (node A1, A3 in Fig. 4.b) are shifted along the boundary to new locations that expand interface element by thickness *t0*.
- For internal node, (node A1, A2 in Fig. 4.c) nodal point is shifted in direction that bisects the directions of two interface elements which the nodal point belong to.
- When the basic mesh include one dimensional bar elements as in Fig. 4d, usually at both side of bar elements interface elements are used. In this case thickness of interface element surrounding bar elements have one half of normal interface element thickness. When expansion takes place, nodal points which belong to bar element are not shifted. Other nodes are moved away in direction normal to bar element direction.

## DRAWING MESH

To check whether element mesh is properly generated, the best way is using graphics representation. For interface of thin layer element cases, it can be done straightforward as in normal two-dimensional meshes. However, in case of zero thickness joint/interface elements, it is very difficult to check if the mesh is plotted in normal way. Interface elements are given an artificial thickness, nodal point co-ordinates are then temporarily modified as in thin layer interface elements and the mesh will be plotted.

## CONCLUSIONS

A flexible method to generate joint / interface elements has been developed. Additional nodal points and interface locations are automatically determined based on their material properties. The method requires very little data even if in complicated domains. It is applicable to all type of one-dimensional and two-dimensional elements. Interface elements for complicated domains such as in geosynthetic or composite material with inclusions statistical distributed can be generated easily. At the contact surface between two different materials, interface elements being inserted or not can be controlled easily. The method can be incorporated into an available automatic meshing program and easily extended into three-dimensional cases.

## REFERENCES

1. Bartune, J., “The automatic generation of the finite element mesh”, *Proceeding of the Sixth International conference on Computer Method and Advances in Geomechanics,* Innsbruck, 11-15 April 1988, Vol. III, pp. 220-2206.

2. Potts, D.M. and Day, R.A., “Automatic mesh generation of zero thickness interface elements”, *Proceeding of the Seventh International conference on Computer Method and Advances in Geomechanics,* 1991, pp. 101-106.

3. Xi, Y., "Representative Volumes of Composite Materials", *Journal of Engineering Mechanics*, ASCE, Vol. 122, No.12 December 1996, pp. 1159-1167.

4. Thai, N.C. and Hasegawa, T., “Prediction nonlinear parameters of soil containing large particles”, *47th National conference of JSIDE*, Kyoto, 1998, pp. 534-535.

5. Desai, C.S., Lightner, J.G. and Siriwardane, H.J., “Thin-layer for interfaces and joints”, *International journal for numerical method in geomechanics*, 1984, Vol. 8, pp. 19-43.