# Automatic Hexahedral Mash Generation for Finite Element Simulation of Metal Forming

포항공대

S.R.Ryoo, S.M.Hwang

(사)한국소성가공학회 2002년도 제 7회 단조심포지움

# AUTOMATIC HEXAHEDRAL MESH GENERATION FOR FINITE ELEMENT SIMULATION OF METAL FORMING

**S.R. Ryoo and S. M. Hwang**
**Department of Mechanical Engineering**
**Pohang University of Science and Technology (POSTECH)**
**San 31, Hyoja-Dong, Nam-Gu, Pohang, 790-784**
**Korea**

## ABSTRACT

A new grid-based approach is presented for automatic generation of hexahedral meshes for simulation of plastic deformation in metal forming. In this approach, special *enveloping* schemes are applied, to eradicate the sources of the degenerate elements that may appear in a generated mesh. The schemes are described in detail, along with a complete procedure for mesh generation. The capability of the approach to deal with an arbitrary, 3-D process geometry is demonstrated through application to a selected forming problem.

## INTRODUCTION

In the past, many researchers proposed various schemes [1-12] for automatically generating either tetrahedral meshes or hexahedral meshes for 3-D process simulation. Among them, a focus may be given to the grid-based approaches as applied to the generation of hexahedral meshes [8-12], considering that an hexahedral mesh can safely accept linear shape functions (which may be a crucial factor in term of reducing the CPU time), and also that the grid-based approaches are inherently applicable to an arbitrary, complex-shaped analysis domain. Unfortunately, existing grid-based approaches lacked a sound methodology to suppress the occurrence of the degenerate elements, and consequently, post processing, such as element splitting or element insertion, had to be performed to cure the degenerate elements, on a case by case basis.

In this paper, a new grid-based approach is presented for automatic generation of a hexahedral mesh in an arbitrarily shaped domain. The approach, which is capable of suppressing the appearance of the degenerate elements, consists of the following steps:

1. Description of the boundary of an analysis domain
2. Construction of a primitive mesh
3. Selection of characteristic nodes
4. Double enveloping
5. Mesh smoothing

Described in detail was each step of mesh generation, with emphasis on the new schemes for the treatment of the degenerate elements. Then, the validity of the proposed approach was demonstrated through application to a selected forming problem.

## MATHEMATICAL REPRESENTATION OF THE BOUNDARY OF AN ANALYSIS DOMAIN

As shown in FIG. 1, the shape of the analysis domain selected for the application of the proposed approach for mesh generation has sharp corners and edges, flat and smooth surface patches, steps, and protruded segments, reflecting a sufficient degree of the shape complexity that the geometry of a workpiece may reveal in industrial forming operations.

The geometrical entities defining the surface of an arbitrary 3-D domain, and therefore should be preserved in the generated mesh, may be classified into *surface patches* and *characteristic* *curves*. A characteristic curve, which represents a sharp boundary line between two adjacent surface patches, may either be closed or open. At a *preform* (initial, undeformed workpiece) stage, an end point of an open characteristic curve is usually a *junction* - the point where two or more characteristic curves meet. However, it is also possible, at a deformed stage, that an end point of a characteristic curve is not a junction but simply an *isolated* end point.

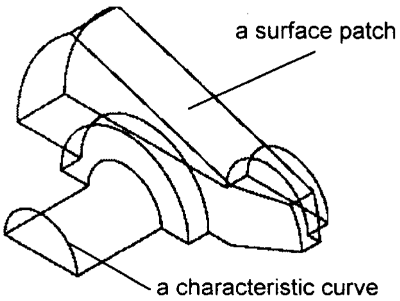
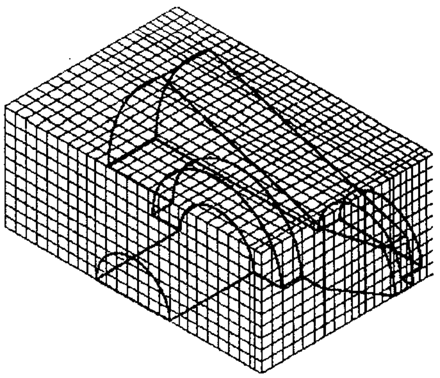


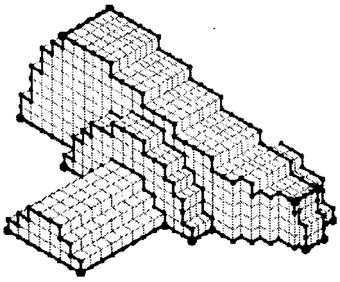
FIG. 1 A WIRE FRAME REPRESENTATION OF THE GEOMETRY OF A WORKPIECE.

## CONSTRUCTION OF A PRIMITIVE MESH

A box-shaped grid is constructed which is comprised of many small brick cells and completely encompassing the analysis domain in its interior, as shown in FIG. 2-(a). Then, the cells are removed from the grid, except those residing entirely in the interior of the domain, resulting in a *primitive mesh* the shape of which, in a rough sense, is similar to the shape of the analysis domain, as shown in FIG. 2-(b). Note that, in a grid-based approach, all of the cells in a primitive mesh are to be transformed into real elements, forming a part of the *final mesh* – the desired mesh representing the complex geometry of the analysis domain.



(a)



(b)

FIG. 2 (A) CONSTRUCTION OF A GRID, (B) A PRIMITIVE MESH.

## SELECTION OF CHARACTERISTIC NODES

The first step of the procedure to transform the primitive mesh into a final mesh is to select, among the nodes on the boundary of the primitive mesh (which may be termed *boundary nodes*), those to be distributed along each characteristic curve (which may be termed *characteristic nodes*).

Since a characteristic curve represents a sharp boundary line, the line between two adjacent characteristic nodes must form an element edge, which may be termed a *characteristic edge*. Therefore, the problem of finding the characteristic nodes for a given characteristic curve is reduced to selecting a series of element edges that are continuous (which may be termed a *path*). Then, each of the nodes that are encountered along a selected path, which may be termed a *characteristic path*, is regarded as a characteristic node.

However, there are many possible paths that can be drawn on the boundary of the primitive mesh. Immediately raised is a problem regarding which path is to be taken as a characteristic path. A criterion employed in this investigation was based on an assumption that a path which is geometrically closest to the characteristic curve is optimal in terms of the quality of the final mesh.

## DEGENERATE ELEMENTS

The final mesh may be obtained by mesh smoothing, which may be performed under the constraints that characteristic nodes must stay on the characteristic curves and that the rest of the boundary nodes must stay on the surface patches. Unfortunately, due to the constraints, *boundary elements* – elements having at least one boundary node, may possibly become degenerate after mesh smoothing. However, the rest of the elements in the primitive mesh, which may be termed *internal elements*, have a full degree of freedom regarding the possible positions that their nodes may occupy and therefore should be free of the degeneracy problem after mesh smoothing. Focusing on the boundary elements, it may be shown that they should belong to one of the following types:

*Type 0 element:*

A Type 0 element is defined as a boundary element having only one *boundary surface* – an element surface located on the boundary of the primitive mesh, and four boundary nodes less than three of which are characteristic nodes. A Type 0 element was illustrated in FIG. 3-(a). Note that the degree of freedom of a Type 0 element is sufficient for preventing the element from becoming degenerate after mesh smoothing(since, among the four boundary nodes at least two nodes can move freely on a surface patch and the rest of the nodes are internal nodes.)

*Type 1 element:*

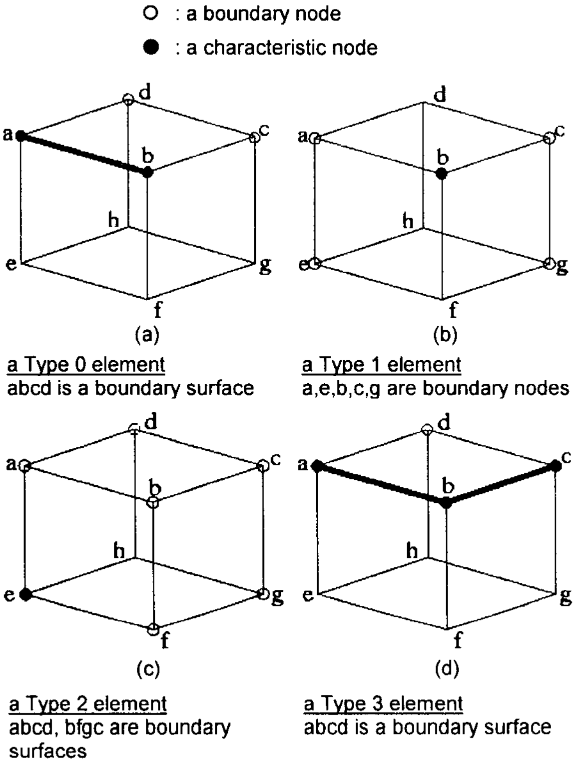
A Type 1 element is defined as a boundary element having at least one boundary node but no boundary surfaces, as illustrated in FIG. 3-(b). Note that the degree of freedom of an element of this type is in general insufficient for preventing the element from becoming degenerate after mesh smoothing (for example, consider an extreme case in which all the eight nodes are characteristic nodes).

*Type 2 element:*

In order for a boundary element to be eligible for a Type 2 element, the element should have at least one boundary surface and at the same time, have at least five boundary nodes, as illustrated in FIG. 3-(c). Note that the degree of freedom is in general insufficient for preventing the element from becoming degenerate after mesh smoothing (for example, consider a case in which three of the six element surfaces are on a flat surface patch).

*Type 3 element:*

In order for a boundary element to be eligible for a Type 3 element, the element should have at least one boundary surface, and among the four nodes on the surface, at least three nodes are characteristic nodes, as illustrated in FIG. 3-(d). It is possible that an element may be of Type 2, and at the same time, of Type 3. In this case, the element is classified as a Type 3 element. Note that the degree of freedom is in general insufficient for preventing the element from becoming degenerate after mesh smoothing (for example, consider a case in which three characteristic nodes on a boundary surface form a straight line).



(a)
a Type 0 element
abcd is a boundary surface

(b)
a Type 1 element
a,e,b,c,g are boundary nodes

(c)
a Type 2 element
abcd, bfgc are boundary surfaces

(d)
a Type 3 element
abcd is a boundary surface

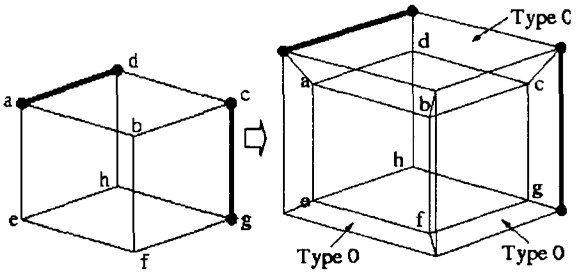
FIG. 3 CLASSIFICATION OF BOUNDARY ELEMENTS.

In summary, a boundary element of any Type, except Type 0, may possibly become degenerate after mesh smoothing. The degeneracy problem may be resolved, either by pre-processing of the boundary elements before mesh smoothing, or by post-processing of the degenerate elements after mesh smoothing. However, an entirely different approach may also be conceived, which was described in detail in the following.

## ENVELOPING, MODE 1 AND MODE 2

We may consider an enveloping operation, in which a new element is created on top of each boundary surface of a boundary element in such a way that the new element has only one boundary surface, which is its top surface. If an original boundary surface has a characteristic node, a new node residing on top of the characteristic node can take over its role. This operation, termed *Mode 1 enveloping*, was illustrated in FIG. 4.

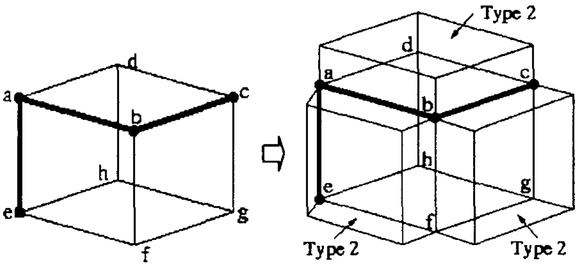
We may consider a different enveloping operation, termed *Mode 2 enveloping*, in which a new element created on top of each original boundary surface is allowed to have five new boundary surfaces. As illustrated in FIG. 5-(a), an original element of any type is transformed into a Type 1 element by this operation, while all the new elements created are exclusively of Type 2.



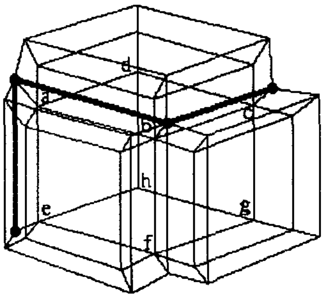
originally a Type 2 element
boundary surfaces are
abcd, aefb, bfgc

after mode 1 enveloping

FIG. 4 MODE 1 ENVELOPING AS APPLIED TO A TYPE 2 ELEMENT.



originally a Type 3 element
abcd, aefb, bfgc are boundary surfaces
(a)



(b)

FIG. 5 (A) MODE 2 ENVELOPING AS APPLIED TO A TYPE 3 ELEMENT, (B) MODE 1 ENVELOPING AS APPLIED TO THE NEW ELEMENTS GENERATED BY MODE 2 ENVELOPING.

## DOUBLE ENVELOPING

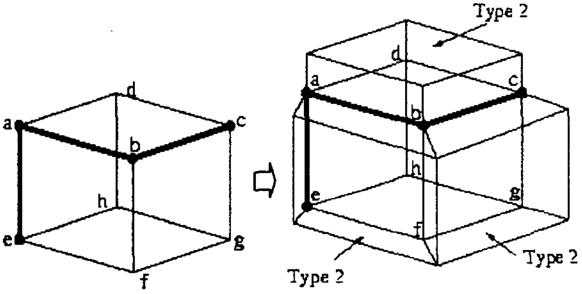
On the basis of the aforementioned characteristics of the two enveloping operations, a straightforward strategy may be developed for the removal of the degenerate elements, as follows:

Step 1: apply Mode 2 enveloping to each original boundary surface in the primitive mesh.
Step 2: apply Mode 1 enveloping to each new boundary surface generated from step 1.
Step 3: relocate the characteristic nodes.

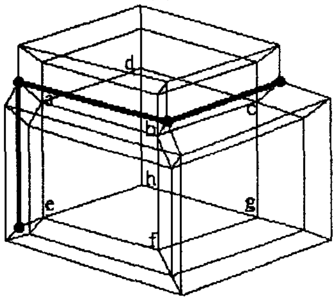
As may be seen from FIG. 5-(b), all the new elements obtained at the completion of step 3 are exclusively of Type 0, demonstrating the validity of double enveloping.

## NON-ISOMORPHIC ENVELOPING

Considering that most of the new elements produced by double enveloping may be irrelevant to the cure of a degenerate element (for example, Mode 1 enveloping, instead of double enveloping, would be sufficient if there are no Type 3 elements), a different operation may be conceived in which the number of the new elements can be minimized, as follows:



originally a Type 3 element
abcd, aefb, bfgc are boundary surfaces
(a)



(b)

FIG. 6 (A) NON-ISOMORPHIC ENVELOPING AS APPLIED TO A TYPE 3 ELEMENT, (B) MODE 1 ENVELOPING AS APPLIED TO THE NEW ELEMENTS GENERATED BY NON-ISOMORPHIC ENVELOPING.

Step 1 - Mode 2 enveloping, is replaced by *Non-isomorphic enveloping*, in which each of five new element surfaces of a new element created on top of an original boundary surface is selectively regarded as a boundary surface, to reduce the number of newly generated boundary surfaces. Recalling that the need for step 1 was raised to cure a Type 3 element, a criterion may be developed for selective exposure of the element surfaces on the basis of information regarding the characteristic edges, as follows: Among the five element surfaces of a new element, only those having a characteristic edge are exposed, plus the top surface. As illustrated in FIG. 6-(a), a new element resulting from non-isomorphic enveloping is either of Type 0 or of Type 2, and therefore, can be accepted as a valid step 1 operation. FIG. 7 partly illustrates the new elements generated from double enveloping as applied to the primitive mesh currently under consideration.

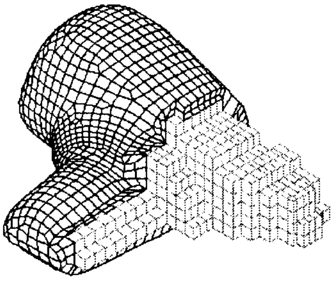


FIG. 7 A PARTIAL ILLUSTRATION OF NEW ELEMENTS GENERATED BY APPLICATION OF DOUBLE ENVELOPING TO THE PRIMITIVE MESH (IN REALITY, THE NEW ELEMENTS CAN NOT BE SEEN AT THIS STAGE, SINCE EACH OF THEM HAS A ZERO VOLUME).

## MESH SMOOTHING

The problem of transforming the primitive mesh modified by double enveloping into a final mesh may be mathematically formulated as a constrained optimization problem, which may be stated as follows:

*design variables:*

$\boldsymbol{X}_i$, $i = 1, 2, \ldots, n$

where $\boldsymbol{X}_i$ is the position vector of $i$th node, and $n$ is the total number of nodes.

*objective function:*

$\phi(\boldsymbol{X}_i)$, $i = 1,2,\ldots,n$

where $\phi$ is a measure describing the mesh quality, the minimum value of which is desired.

*design constraints:*

$\|\boldsymbol{X}_k - \boldsymbol{Y}_k\| = 0$, $k = 1,2,\ldots, m$

where $k$ denotes a boundary node, and $\boldsymbol{Y}_k$ denotes the position vector of a point either on a surface patch or on a characteristic curve (if $k$ is a characteristic node) which is closest to the node $k$.

The present problem may be solved by a *sequential unconstrained optimization technique* [13], in which the problem is reformulated as an unconstrained optimization problem with a modified objective function

$$\psi(\boldsymbol{X}_k) = \phi(\boldsymbol{X}_k) + \xi \sum_{k=1}^{m} \|\boldsymbol{X}_k - \boldsymbol{Y}_k\| \qquad (1)$$

where $\xi$ is a penalty constant, the value of which is to be gradually increased as iteration for optimization is continued. Note that $\boldsymbol{Y}_k$ is not a prescribed vector but varies with the current nodal position vector $\boldsymbol{X}_k$.

There are many possible choices in selecting an objective function, as may be found in the references [14,15]. In this investigation, an objective function was generated on the basis of a measure proposed by Oddy [16]. The optimized mesh resulting from the design iteration is shown in FIG. 8.

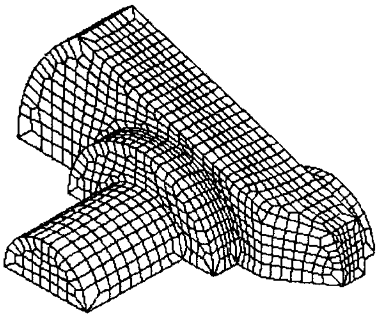


FIG. 8 A FINAL MESH.

## RESULTS AND DISCUSSION

The problem investigated was forging of a cylindrical bar into a crank shaft part. Due to the symmetry of the part, only one-quarter of the process geometry was considered, as shown in

FIG. 9. As shown in FIG. 10 and in FIG. 11, mesh generation had to be performed frequently (17 times in total), even at an early stage of forming, due to the complex nature of the die cavities to fill. The characteristic curves that appeared during process simulation were in general very complex, some of which having an isolated point, but mesh generation was successfully accomplished, as illustrated in FIG. 12, indicating the validity of the present approach for simulation of 3-D plastic deformation in forming of an arbitrary shaped part.

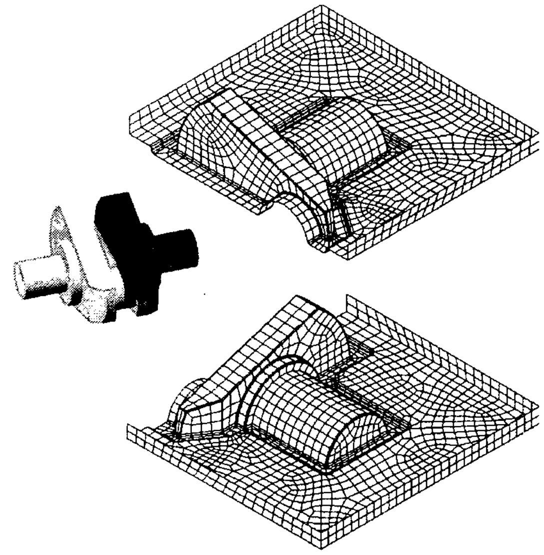


FIG. 9 THE SHAPE OF A DIE USED IN FORGING OF A CRANK SHAFT.

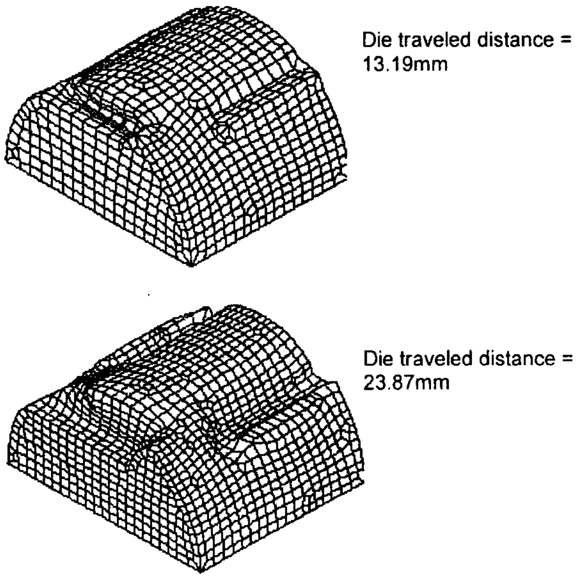


FIG. 10 INTERMEDIATE SHAPES OF THE WORKPIECE IN FORGING OF A CRANK SHAFT (AFTER MESH GENERATION).

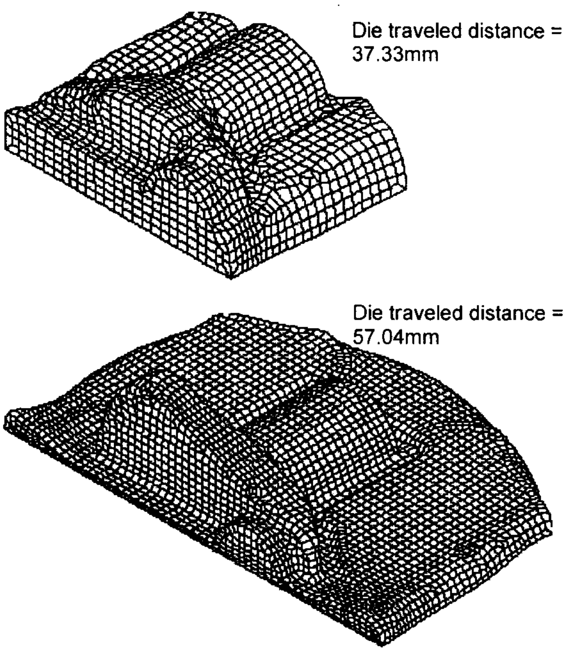


FIG. 11 INTERMEDIATE SHAPES OF THE WORKPIECE IN FORGING OF A CRANK SHAFT (CONTINUED) (AFTER MESH GENERATION).

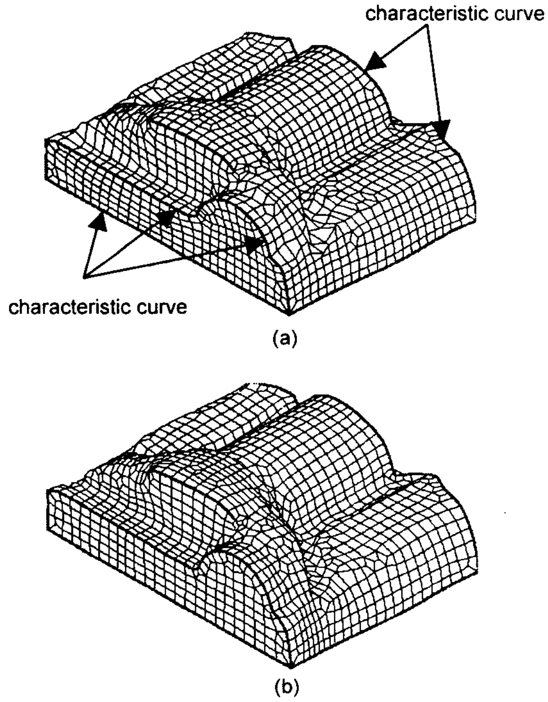


FIG. 12 (A) A DISTORTED MESH, (B) AFTER MESH GENERATION(DIE TRAVELED DISTANCE = 37.33MM).

## CONCLUDING REMARKS

It was demonstrated through this paper that the present approach can be successfully applied to process simulation in metal forming involving an arbitrary shaped part. A main merit of the present approach lies in that, by completely eradicating the sources of the degeneracy that may possibly occur in a generated mesh, simulation may be performed without any human intervention, which may become extremely difficult depending upon the process geometry to be dealt with.

## ACKNOWLEDGEMENTS

The authors wish to thank Korean Science and Engineering Foundation for their financial support (Grant no. 971-1007-004-2), which made the present investigation possible.

## REFERENCES

1. Rassineux A.,(1997) "3D mesh adaptation. Optimization of Tetrahedral meshes by advancing front technique," *Computer Methods in Applied Mechanics and Engineering*, Vol. 141, pp.335-354.

2. Hattangady N.V.,(1999), "Automatic remeshing in 3-D analysis of forming processes," *International Journal for Numerical Methods in Engineering*, Vol. 45, pp. 553-568.

3. Blacker T.D., Meyers R.J.,(1993), "Seams and wedges in plastering: A 3-D hexahedral mesh generation algorithm," *Engineering with Computers*, Vol. 9, pp. 83-93.

4. Tam T.K.H., Armstrong C.G.,(1993), "Finite element mesh control by integer programming," *International Journal for Numerical Methods in Engineering*, Vol. 36, pp. 2581-2605.

5. Li T.S., Mckeag R.M., Armstrong C.G.,(1995), "Hexahedral meshing using midpoint subdivision and integer programming," *Computer Methods in Applied Mechanics and Engineering*, Vol. 124, pp. 171-193.

6. Price M.A., Armstrong C.G.,(1995), "Hexahedral mesh generation by medial surface subdivision: Part I. Solids with convex edges," *International Journal for Numerical Methods in Engineering*, Vol. 38, pp. 3335-3359.

7. Price M.A., Armstrong C.G.,(1997), "Hexahedral mesh generation by media 1 surface subdivision: Part II. Solids with flat and concave edges," *International Journal for Numerical Methods in Engineering*, Vol. 40, pp. 111-136.

8. Schneiders R., Oberschelp W,(1992), "New and effective remeshing scheme for the simulation of metal forming processes," *Engineering with Computers*, Vol. 8, pp. 163-176.

9. Schneiders R., Bunten R,(1995), "Automatic generation of hexahedral finite element meshes," *Computer Aided Geometric Design*, Vol. 12, pp. 693-707.

10. Schneiders R,(1996), "A grid-based algorithm for the generation of hexahedral element meshes," *Engineering with Computers*, Vol. 12, pp. 168-17.

11. Tekkaya A.E., Kavakli S.,(1995), "3-D simulation of metal forming processes with automatic mesh generation," *Steel Research*, Vol. 66(9), pp. 377-383.

12. Lee Y.K., Yang D.Y.,(1999) "Development of a grid-based mesh generation technique and its application to remeshing during the finite element simulation of a metal forming process," *Engineering with Computers*, Vol. 16, pp. 316-336.

13. Haug E., Arora J.S.,(1973), *Applied Optimal Design - Mechanical and Structural Systems*. Wiley, New York.

14. Knupp P.M.,(2000), "Achieving finite element mesh quality via optimization of the Jacobian matrix norm and associated quantities. Part I – a frame work for surface mesh optimization," *International Journal for Numerical Methods in Engineering*, Vol. 48, pp. 401-420.

15. Knupp P.M.,(2000), "Achieving finite element mesh quality via optimization of the Jacobian matrix norm and associated quantities. Part II – a frame work for volume mesh optimization and the condition number of the Jacobian matrix," *International Journal for Numerical Methods in Engineering*, Vol. 48, pp. 1165-1185.

16. Oddy A., Goldak J., Mcdill M., Bibby M,(1988), "A distortion metric for isoparametric finite elements," *Transactions of the CSME*, Vol. 12(4), pp. 213-217.