[Mesh Generation]
Computational Geometry (CSc 635)

Jagdish Bhatta

Central Department of Computer Science & Information Technology
Tribhuvan University

Mesh

(Transferring continuous space into discrete space for easier calculation or Process of transferring continuous models into discrete counter parts)

Mesh is the complex of elements ***discretizing*** the simulation domain, e.g. Triangular or quadrilateral mesh in 2D, tetrahedral or hexahedral mesh in 3D Thus, meshing can be defined as the process of splitting the physical domain to smaller sub-domains, with meshing surface domains may be divided into triangular or quadrilateral shapes & the volumes can be subdivided into tetrahedral or hexahedral shapes. In general, meshes are used to model the objects in computer graphics. The high quality graphics system typically model objects with polygon meshes as the surface of any object can be represented by the polygonal faces.

The most basic form of mesh classification is based upon the connectivity of the mesh; structured or unstructured.

Mesh Types [By Configuration]:

(1) Structured Meshes

If the internal nodes in the mesh are surrounded by a constant number of elements, such a mesh is called a structured mesh. Thus, a structured mesh is characterized by regular connectivity of the elements of mesh.

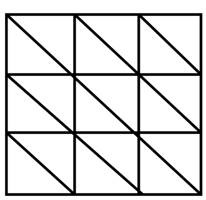


Fig: Each internal node connected with 6 elements

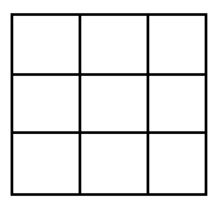


Fig: Each internal node connected with 4 elements.

The structured meshes are simple and efficient in numerical computations. A structured mesh requires less memory than an unstructured with same number of elements because array storage can define neighbor connectivity implicitly. Structured grid calculation takes less time than an unstructured grid calculation. The major advantage of structured mesh is having their compatibility with efficient finite difference algorithms. However, the structured meshes are not suitable for complex geometric shapes as it can be difficult to compute the structured mesh for complicated geometric domain. Further in structured mesh, it may require more elements than in unstructured mesh because no. of elements in structured mesh is of fixed size while the size may be upgraded in unstructured meshes.

(2) Unstructured Mesh:

If internal nodes in the mesh are not necessarily surrounded by a constant number of elements, then it is unstructured mesh. Thus, in such meshes, there may be any number of elements meeting at a single node.

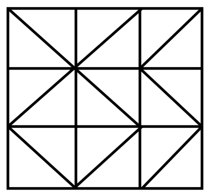


Fig: Unstructured Mesh

Delaunay triangulation is one of the approaches for construction of unstructured meshes. The advantage of using unstructured mesh is its flexibility in fitting complicated domains so they are good choice for constructing complex objects.

With unstructured mesh, the refinement of mesh is easy as there we have rapid grading from small to large elements. However, the unstructured mesh requires more memory than the structured mesh with same no. of elements. Also, the unstructured meshes are expensive in calculation/computation time. Unstructured meshes are much more difficult to generate as there we may have variation of elements size.

(3) Hybrid mesh:

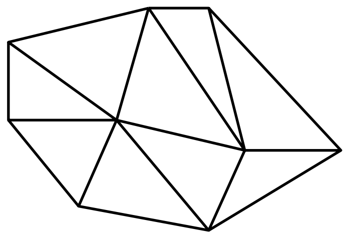
A hybrid mesh is formed by a number of structured meshes arranged in an overall unstructured pattern. Hybrid mesh falls in between structured & unstructured meshes. These meshes are used in the problems having complicated geometric objects.

Mesh Types [by Elements]:

According as the types of elements it consists of, mesh can also be categorized as, Triangle/Tetrahedral mesh and Quad/Hexahedral mesh, each of which are considered in two dimensions/ three dimension respectively.

(1) Triangle/Tetrahedral meshes:

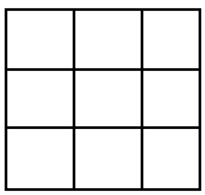
Here, the surface domains are subdivided into triangular elements, while the volumes in 3D are subdivided into tetrahedral elements.



These types of mesh are most common form of unstructured mesh generation. Techniques like Delaunay triangulation, quad tree approach are used for generation of such of meshes.

(2) Quad/Hexahedral Meshes:

With these types of meshes, the two dimensional surface domains can be subdivided into quadrilateral elements while 3D volumes can be subdivided into hexahedral elements.



Both structured & unstructured meshing methods can be used for generating such meshes.

Mesh Validity:

A mesh is said to be the valid mesh if it has following properties:

- There are no holes in the mesh.
- There are no intersecting elements in the mesh.
- The elements are well shapes.
  - Aspect ratio is close to 1, as much as much as possible.
  - Edge angles are close to $\pi/2$, as much as possible
- The boundary nodes are located at the object boundary.

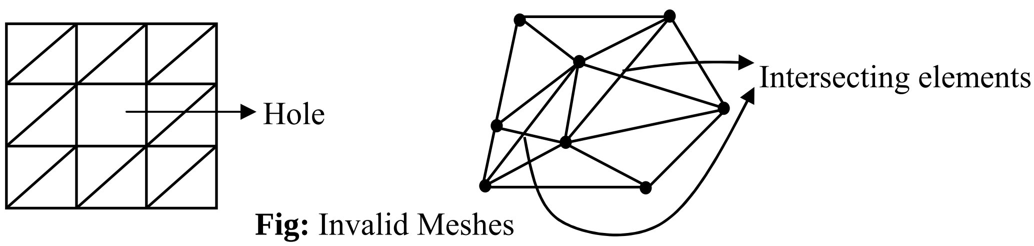


Fig: Invalid Meshes

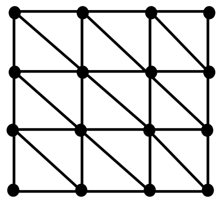


Fig: Valid Mesh

Criteria for a good Meshing:

Shape of elements

- A good quality mesh should avoid both very sharp and flat angles.

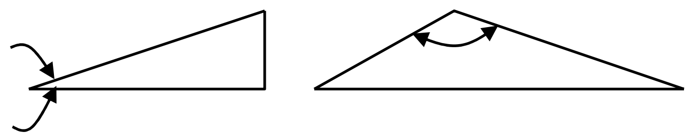


Fig: - Elements with sharp & Flat angles.

Number of elements

- The number of elements in a good mesh should be moderate; related to efficiently of finite element analysis.

Topological consistency

- A good mesh has topological consistency or homeomorphism between exact input domain & its mesh.

Adaptability, automation, conforming mesh, etc

Mesh Conformity: -

Conformity of a mesh is determined by the adjacent elements within the mesh. If the adjacent elements share a vertex or whole edge, the mesh is conforming otherwise the mesh is non conforming.

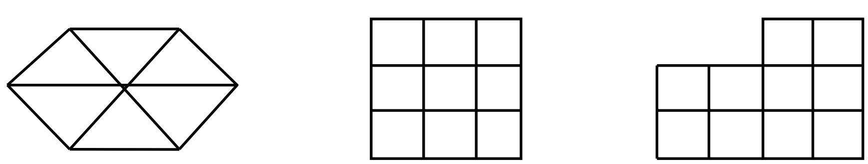


Fig: - Confirming meshes

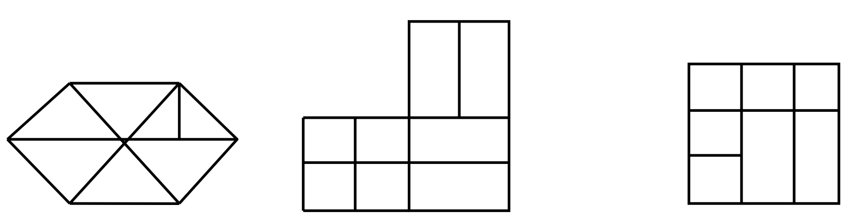


Fig: Non-confirming meshes

Mesh Conversion: -

If a mesh generator produces only one type of meshes and our requirement is of another type, then mesh conversion is needed. Examples of mesh conversion may be such as from quadrilateral mesh to triangular mesh, from triangular mesh to quadrilateral mesh.

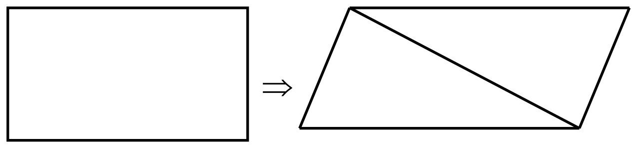


Fig: Quadrilateral ⇒ Triangular mesh (No new node is inserted)

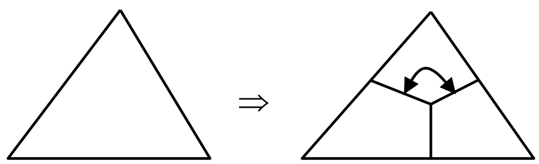
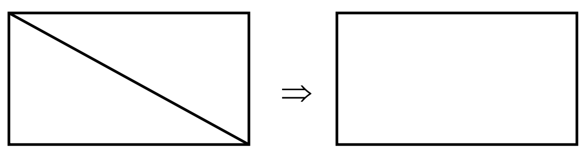


Fig: Triangular Mesh ⇒ Quadrilateral mesh

Here, new node is inserted during conversion; however elements will not be well shaped due to the flat angles.

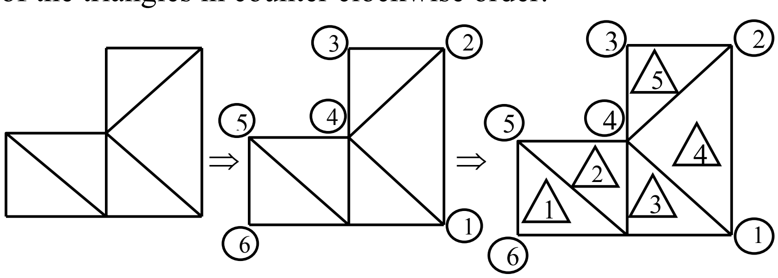


Global Vertex Numbering:

In order to represent the topology of triangular elements in the mesh, a concept of global vertex numbering is used with this idea; we create a global numbering of unique vertices in the mesh. The allocation of the numbers to each vertex is arbitrary.

Thus, label each triangle in mesh. The labeling of the triangles is done in arbitrary order.

Now we can represent each triangle as a triplet of integers which are global vertex numbers of the triangles in counter clockwise order.



Triangle 1: 6, 7, 5
Triangle 2: 7, 4, 5
Triangle 3: 7, 1, 4
Triangle 4: 1, 2, 4
Triangle 5: 2, 3, 4

Mesh Representation:

A mesh consists of a set of vertices, edges & faces. Thus mesh representation is done through proper use of data structures that defines how each of the elements is stored and they are referenced. To represent the meshes *face-based* and *edge-based* data structures can be used.

- *Face-based data structures:* such type of data structures store each face pointers to its vertices and its neighboring faces. Thus, we can navigate through each vertex by visiting all surrounding faces.

- *Edge-based data structures:* such type of data structures store pointers to each vertex of an edge & its neighboring edge. In general, as meshes are planar geometric graph so doubly connected edge list is one of the efficient data structure for representing meshes.

Mesh Generation (Grid generation):

Mesh generation refers to the practice of generation a polygonal or polyhedral mesh that approximates a geometric domain. Mesh generation plays a vital role in computational field simulation for high performance computing. The mesh can tremendously influence the accuracy & efficiency of a simulation. There are different methods for generating structured and unstructured meshes.

The structured meshes can be generated using "*bottom up*" or "*mapped*" approaches.

***Bottom up approach:* -** In this method, it works in bottom up manner. Here, the vertices are first meshed, followed by curves, then surfaces are meshed and finally the solids

- Here, the input for the subsequent meshing operation is the result of the previous lower dimension.
- At first, nodes are distributed along geometric curves. The result of curve meshing process provides input to a surface meshing, where a set of curves define a closed set of surface loops the next step is to decompose the surface into fine mesh elements as triangles or quadrilaterals. The set of surface areas are then used as input for volume mesher for the formulation of tetrahedral or hexahedral mesh.

***Mapped Mesh Generation:* -** In this mesh generation approach; mesh is generated by mapping the physical domain into computational space. This approach is an earliest method for mesh generation appeared in 1970's.

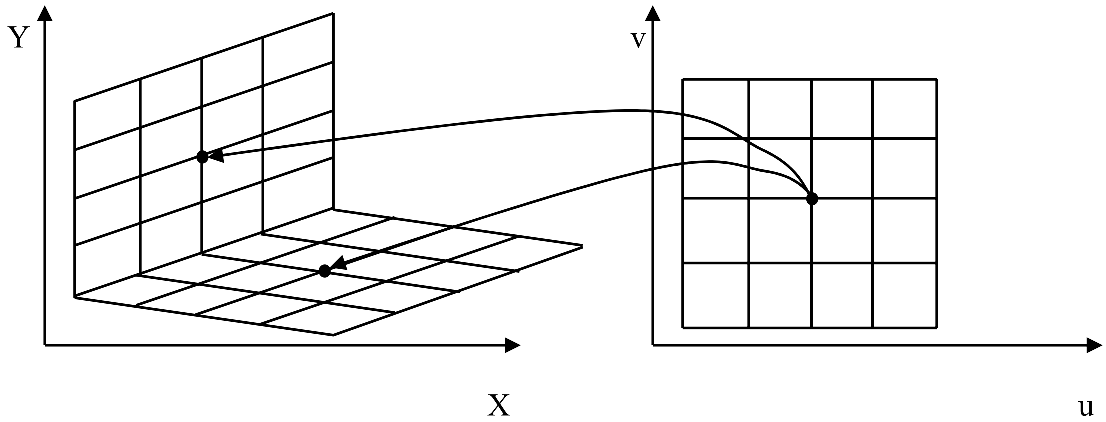


Fig (1) Mapped element approach

Here, first subdivide the physical domain into simple regions. E.g. quadrilaterals, triangles

- Generate mesh in computational space (u, v) as in fig (1).
- Determine nodes in physical domain using shape functions.

There are different approaches for unstructured mesh generation such as;

- Topology/Geometric decomposition methods
- Delaunay triangulation based method
- Quad tree based method.

Topology/Geometric Decomposition Methods

a) Topology decomposition approach:

Here, the idea is to repeatedly subtract a simplex form of an object from the given geometric object until the given object is itself reduced to a single simplex object.

- In case of 2D, the simplex form of object is triangle. So for meshing cut off the triangles whose vertices are the vertices of given object until only 3 vertices are left.

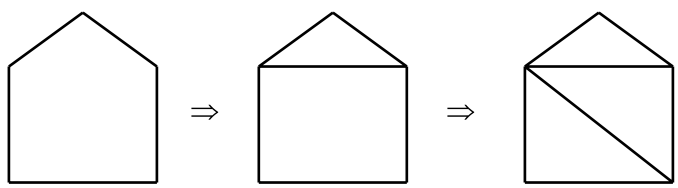
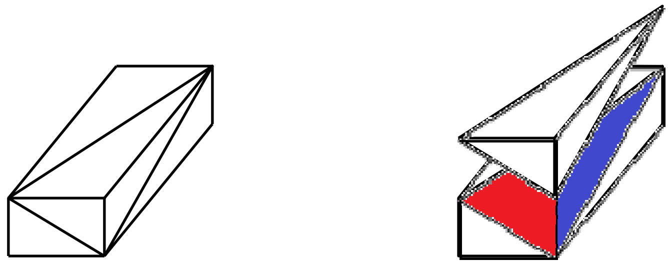
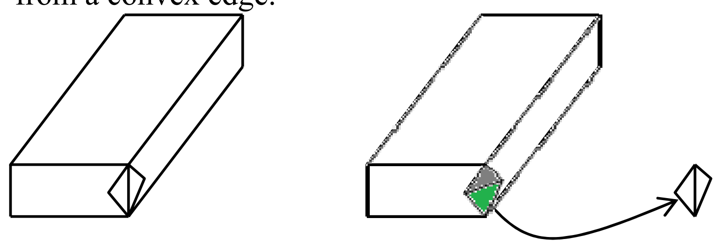


Fig: 2D topology Decomposition

- In case of 3D, simplex form of object is tetrahedron, so each time; a tetrahedron is removed from the object until it reduces to a tetrahedron itself. So construct a tetrahedron by cutting out a corner vertex which has exactly 3 adjacent edges.



- If all remaining vertices have more than three adjacent edges then dig out tetrahedron from a convex edge.



Following above steps, a 3D object can be meshed into a number of tetrahedrons.

 # This topology decomposition approach generates the gross/coarse elements (non refined), so the mesh needs to be refined later to fulfill the required mesh density distribution.

b) Geometric Decomposition Approach: -

- **Recursive Subdivision:**

The recursive version of geometric decomposition approach consists of following steps:

- ➢ Start with a convex object
- ➢ Insert nodes in the boundary of the object according to the mesh density distribution.
- ➢ Divide the object roughly in the middle of its longest axis.
- ➢ Insert nodes into the split line according to mesh density distribution.
- ➢ Recursively divide the two halves until they become triangles.

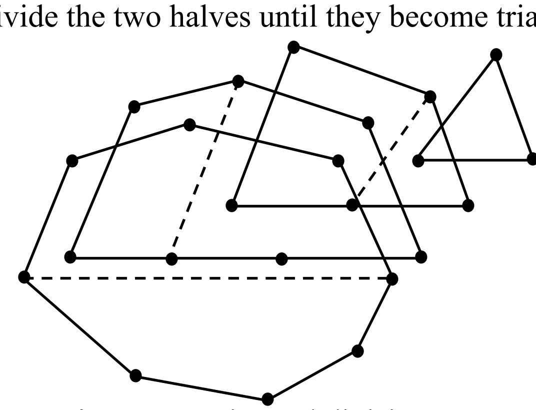
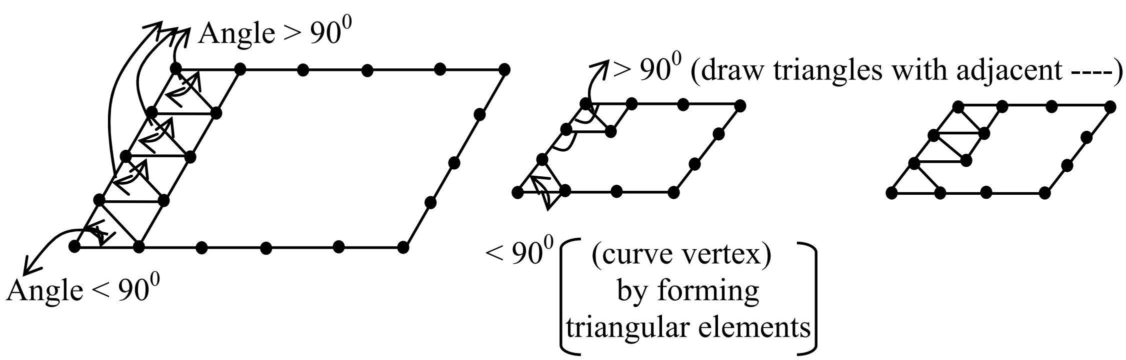


Fig: - Recursive Subdivision

- **Iterative Subdivision:**

  This is another variation of geometric decomposition and consists of iterative removal of elements. It consists of following steps;

  - Start with a simply connected region,
  - Insert nodes into boundary of the object to satisfy mesh density distribution.
  - Remove all vertex angles of the polygon which are less than 90^0 by forming triangular elements.
  - For the vertices with an angle of less than 180^0 but above 90^0, form two triangles with adjacent nodes to the vertex.
  - Repeat above two steps until last three points form a triangular element.

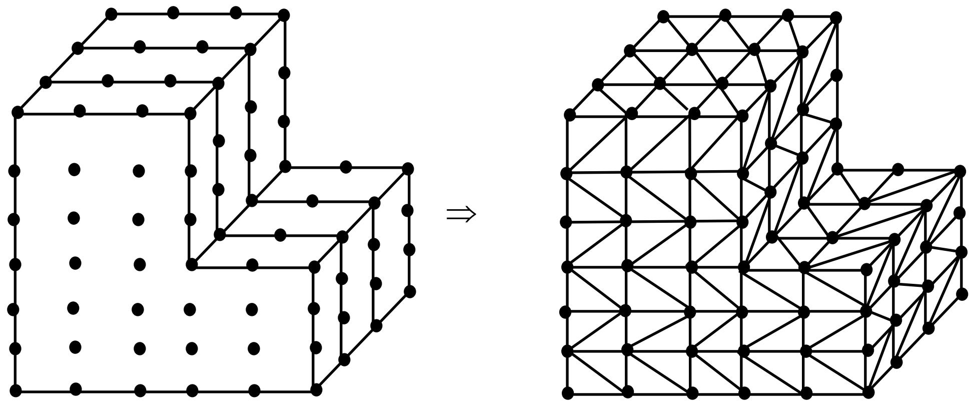


- **Delaunay Triangulation Based Method:**

  - Delaunay triangulation is constructed by connecting those points whose Voronoi regions have a common edge in 2D or common face in 3D; in other words, if Delaunay criterion is satisfied. The Delaunay criterion is sometimes called “empty circle” or “empty sphere” property in 2D & 3D respectively.

  - Delaunay triangulation can be used for meshing the given geometric object. The Delaunay criterion provides the criteria for which to connect a set of existing point in the space.

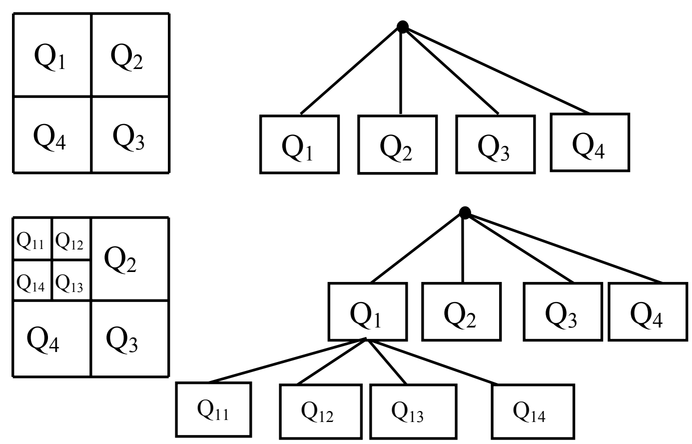
  - At first mesh the boundary of the geometric object to provide an initial set of nodes. The boundary nodes are then triangulated according to the Delaunay criterion. Nodes are then inserted incrementally into the existing mesh, redefining the triangles or tetrahedral locally as each new node is inserted.



Meshed object (Delaunay triangulation of points)

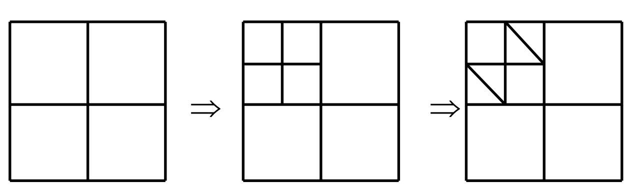
- **Quad Tree Method:**

  This quad tree method is used for meshing the 2D-space. It comprises the decomposition of 2D space into a number of quadrants (usually the squares). Each node in the quad tree has four data elements, one for each quad rants in the region of representation.



This method consists of following steps:

- Subdivide the object interior into quadrants whose size satisfies the mesh density distribution.
- Finally, each quadrant is broken up into triangles such that resulting mesh is conforming.



The mesh generated by quad tree decomposition could result excellent interior but not well shaped boundary elements, because the quadrants on boundary may have cut corners.

Mesh Refinement:

The meshes generated from different meshing algorithms may not be satisfying the required mesh density. There may be chance of gross mesh elements that do not have proper uniformity. So, to satisfy the required mesh density, the mesh needs to be refined after its generation.

Mesh refinement includes increasing mesh density from region to region satisfying the required density. Following, we have, one approach for refinement of mesh;

Let T_0 be any conforming initial triangulation having N_0 triangles. Then a new triangulation Ti is defined as,

1) Bisect triangles t by the largest side for all $t \in T_0$

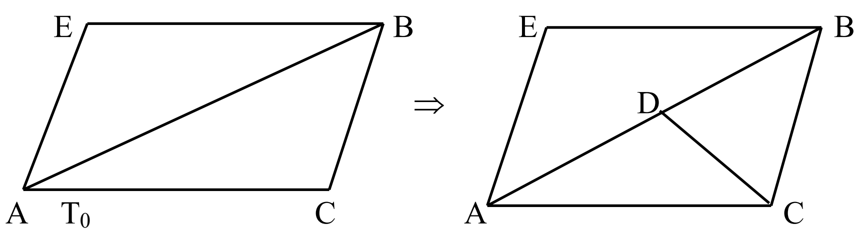


Fig (1) Refinement of Mesh

2) Find the set S1 of triangles generated in step 1 & such that the midpoint of one of its side is a non-conforming node. E.g. S1 = {ΔABE}, Where D is non-conforming node.
3) For each triangle $t \in$. join its non conforming node with opposite vertex; i.e. join D with E. This results conforming triangulation.
4) Proceed likewise so as to meet the required mesh density.

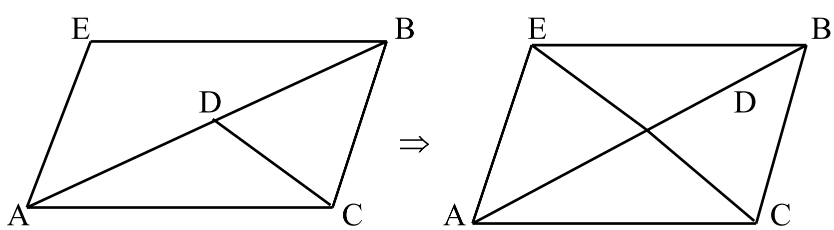


Fig (2) Refinement of Mesh