In The Name of God

Shahed University
Faculty of Engineering
Department of Civil Engineering

The Substructuring Technique in finite element method in 2D problems

User Guide

By:
Hadi kadkhodaei
Hadi.kadkhodaei@yahoo.com

Table of Contents

Introduction

The accurate analysis and design of large and complex structures remains a challenging task for engineers. Major advances in fast computing technologies have encouraged engineers to consider more complex constitutive models in analysis of structures. The finite element method (FEM) is the most common method used and has played a key role in the development.

Engineers are increasingly interested in accurate analysis and consideration of the nonlinear condition, large deformations, and cases where the approximation is reduced. Where large and highly-complex structures are involved, analysis can take hours and even days. Software producers continually endeavor to reduce analysis time of complex structures. One method of reducing the amount of computation is the technique of substructuring in which a large structure is subdivided into smaller parts that can be analyzed separately . For example If we separate the aircraft into substructures, such as parts of the fuselage or body, wing sections, and so on, as shown in Figure 5ـ31(b), then we can solve the problem more readily and on computers with limited memory.

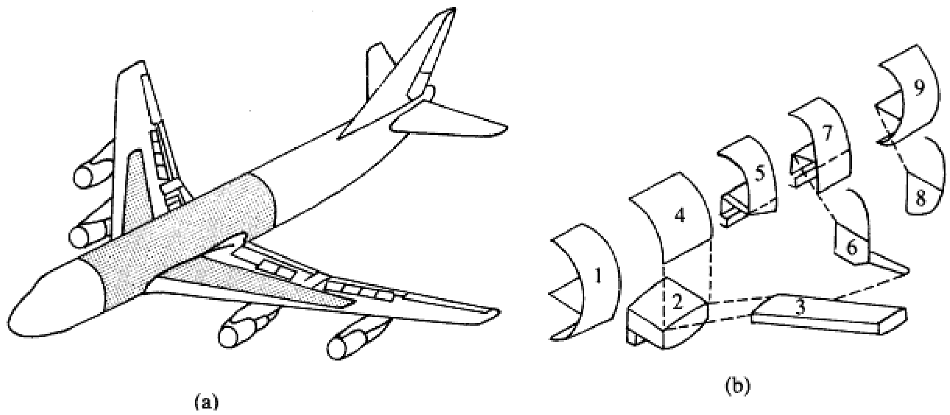


Figure 1 Airplane frame showing substructuring. (a) Boeing 747 aircraft (shaded area indicates portion of the airframe analyzed by finite element method). (b) Substructures for finite element analysis of shaded region

Substructure method

In this technique, the structure is divided into substructures with each substructure containing several elements. The degrees of freedom (DOF) of a substructure are classified as:

- Internal DOF: not connected to the DOF of any other substructure
- Boundary DOF: connected to at least one other substructure; these usually reside at the *boundary nodes* placed on the periphery of the substructure

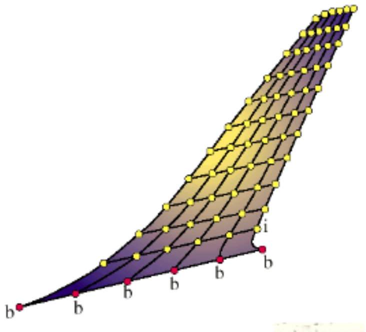


Figure 2(bounary & internal nodes)

If the equilibrium equation is written in boundary DOFs, the objective will be to eliminate all displacement DOF associated with *internal freedoms*. This elimination process is called *static condensation* or, simply, *condensation.* The static condensation method assumes that those internal DOF that can be condensed are arranged in the first *i* nodal coordinates and the remaining boundary DOF are the last *b* nodal coordinates. This arrangement allows the governing equation for a structure to be written using partitioned matrices as:

$$\begin{bmatrix} [K]_{ii} & [K]_{ib} \\ [K]_{bi} & [K]_{bb} \end{bmatrix} \begin{Bmatrix} \{u_i\} \\ \{u_b\} \end{Bmatrix} = \begin{Bmatrix} \{F_i\} \\ \{F_b\} \end{Bmatrix} \tag{1}$$

where subscripts *i* and *b* represent the internal and boundary DOF, respectively. A simple multiplication of the partitioned system in Eq. (1) yields the following two matrix equations:

$$[K]_{ii}\{u\}_i + [K]_{ib}\{u\}_b = \{F\}_i \tag{2}$$

$$[K]_{bi}\{u\}_i + [K]_{bb}\{u\}_b = \{F\}_b \tag{3}$$

Solving Eq. (2) for $\{u\}_i$ and substituting it into Eq. (3) arrives at:

$$\{u\}_i = [K]_{ii}^{-1}\left(\{F\}_i - [K]_{ib}\{u\}_b\right) \tag{4}$$

$$\left(\{F\}_b - [K]_{bi}[K]_{ii}^{-1}\{F\}_i\right) = \left([K]_{bb} - [K]_{bi}[K]_{ii}^{-1}[K]_{ib}\right)\{u\}_b \tag{5}$$

Eq. (5) may be written as:

$$\left\{\overline{F}\right\} = \left[\overline{K}\right]\left\{u\right\}_b \tag{6}$$

in which:

$$\left\{\overline{F}\right\} = \left\{\overline{F_b}\right\} - \left[K\right]_{bi}\left[K\right]_{ii}^{-1}\left\{F\right\}_i \tag{7}$$

and:

$$\left[\overline{K}\right] = \left[K\right]_{bb} - \left[K\right]_{bi}\left[K\right]_{ii}^{-1}\left[K\right]_{ib} \tag{8}$$

In Eqs. (7) and (8), $\left\{\overline{F}\right\},\left[\overline{K}\right]$ are the condensed stiffness matrix and force vector, respectively. This technique produces a condensed stiffness matrix and a condensed force vector for each substructure which are associated only with the boundary DOF. Assume that each substructure is equivalent to an element having stiffness matrix and nodal force $\left[\overline{K}\right]$and $\left\{\overline{F}\right\}$, respectively. Classic FEM states that the condensed equations of substructures must be assembled to obtain the condensed governing equation of the whole structure for the total boundary DOF as:

$$\left[\overline{K}\right]_t\left\{u\right\}_{b_t} = \left\{\overline{F}\right\}_t \tag{9}$$

in which$\left[\overline{K}\right]_t$, $\left\{\overline{F}\right\}_t$ are the assembled $\left[\overline{K}\right]and\left\{\overline{F}\right\}$, respectively.

This obtains the solution to the Eq. (9) boundary DOF. By substituting boundary displacements associated with each substructure into Eq. (4), the internal displacements in each substructure are computed.

Matlab code

For solving finite element problems with substructure method , substructure2_4 is prepared in matlab code , in wich many subfunctions are involved to compute the analyse. Also for compare finite element method with substructuing , finite_element code is exist in code package . these codes are designed for 2D problems with elements of 4 node. This package is prepared in shahed university. all right reserved for shahed university.

analysis capabilities

problems with structured meshing

For structured meshes in 2D problems, the number of elements in each direction are assumed to be m and n. The number of substructures in each direction are denoted as na and nb. Adjacent substructures are the same size. The partition of substructures is considered on a regular basis

Problems arbitrary mesh and a separate region for substructuring

In this kind of problem we are faced with a mesh that is divided into regions, each of which has its own separate subdivision. this case was general and we can define any type of substrcturing with arbitrary subdivisions.
For this purpose, two examples are given for a better understanding of working with the this software.

Example1:Rectangular plate with hole

this example is a rectangular plate with a central circular hole that is tensioned from opposite sides. Only half of the problem was modeled for symmetry, as shown in Figure 3. The plate was meshed by 30□30 structured 4-noded quadrilateral elements. the geometry & meshing of problems is as follows. Loading in each related node is 100 N. the poisson's ratio is 0.3 and the Young's modulus is 2E+11 . each direction of the meshing is divided to 5 section and total number of substructure is 25.

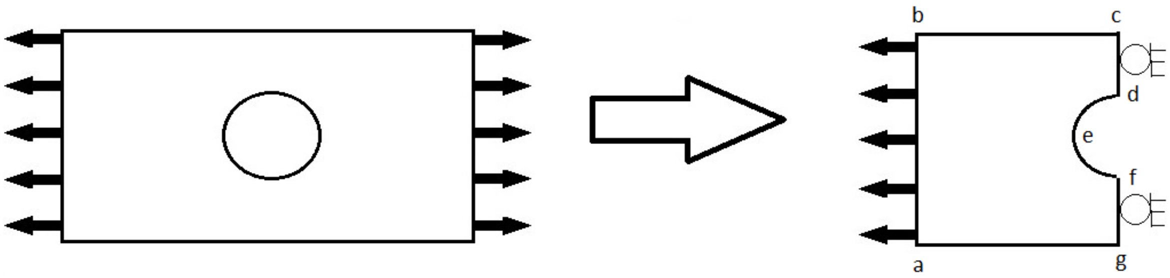


Figure 3(geometry of example 1)

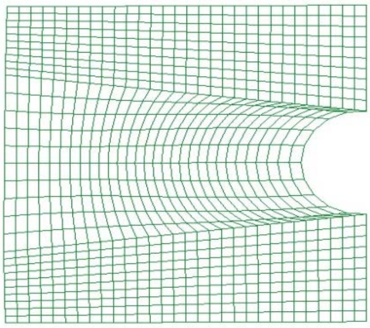


Figure 4(meshing of example 1)

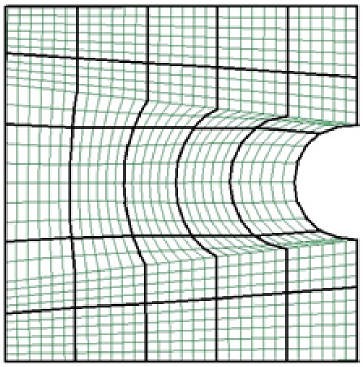


Figure 5(substructuring of example 1)

Required Information about General Variables

nu- Poisson's ratio.
E- Young's modulus.

Sublevel - sublevel is an array of all ones. Size of this array is equal to the number of substructures. The value For those substructure that obtaining internal displacement is not necessary is zero. This array by default is one for all substructures.

Auto_sub - this valuable is one by default. It's means that meshing of geometry is structured and substructure has equal size of element in each direction. The zero value means that substructuring is arbitrary and should be read file region in main directory of input_data.

Auto_coord - this valuable is zero by default. It's means that the coordinate of nodes should be read from "node.txt" in main directory. The zero value is used If we want rectangular geometry for software test.

Auto_elem - this valuable is zero by default. It's means that the coordinate of nodes should be read from "node.txt" in main directory.

Manual_rest - this valuable is zero by default. It's means that the coordinate of nodes should be read from "restrain.txt" in main directory.

Condition - if this value is one, internal displacement of all substructures is needed and the type of analyse is free analyse. Planecondition -if it is one, the problem is plane stress. Else if the value is zero, the problem is plane strain.

Keynode - each structured meshing has 4 edge that Could be the basis for regular element numbering. This valuable create automatically in the software, but because of coincidenc with our definition of numbering, Recommended to be initialized by user.
na4,nb4 - these valuables are the number of subdivision in each direction for the case that meshing is structured and we donب define sub structure in region folder from main directory.

St_add - this valuables is the address of main directory from witch all input data read. user must input all of the files that are read in the program in this patch(for example "node.txt","elem.txt" etc)

```
st_add='C:\Users\Mahak110\Desktop\modify\input_data\data5';
manual_sublevel=0;
auto_sub=1;      auto_coord=0;     auto_elem=0;
manual_f=0;      manual_rest=0;
nu=0.3; planecondition=1; E=2.0000e+011;
condition=1;
keynode=961;
na4=5;   nb4=5;
```

Figure 6(General input data)

Required Information about Geometry

Coordinate of Nodes

Coordinate of each node is contain in "node.txt". as Figure 7, first Column is number of nodes, second and third Column is coordinate of nodes.(x & y respectively)

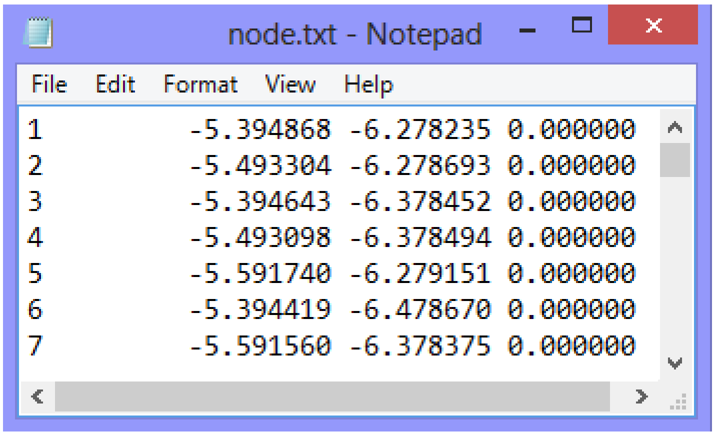
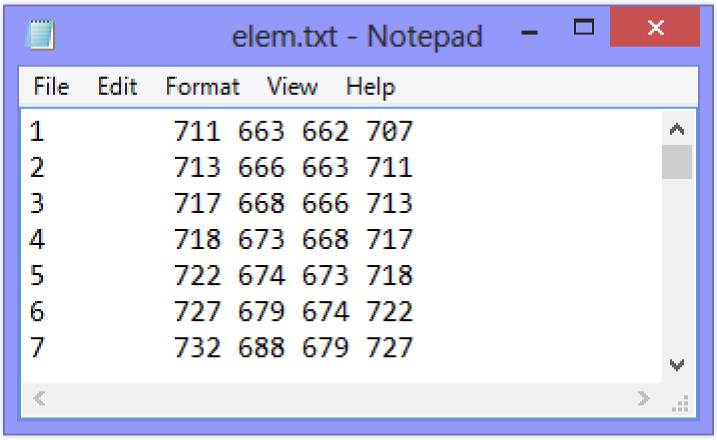


Figure 7(Input Procedure for Coordinate of Nodes)

Elements Information

Each element is defined by four points Therefore, to specify an element the element nodes with their number is required. Elements information is contained in "elem.txt". as Figure 8, first Column is element number, 4 subsequent Column is node number that sorted counter clockwise

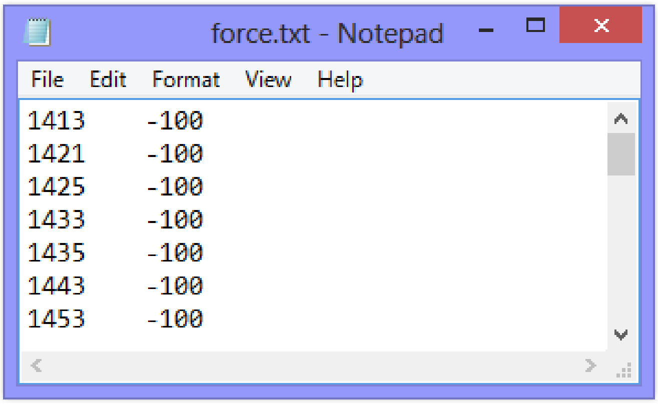


1	711	663	662	707
2	713	666	663	711
3	717	668	666	713
4	718	673	668	717
5	722	674	673	718
6	727	679	674	722
7	732	688	679	727

Figure 8(Input Procedure for Elements Information)

Loading Information

Some degrees of freedom(Dof) were applied by loading .these information should be written in "force.txt". as The first column is Dof number and the second is related force.

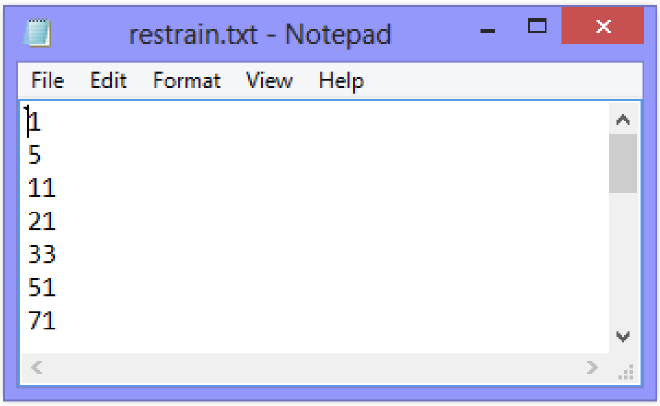


1413	-100
1421	-100
1425	-100
1433	-100
1435	-100
1443	-100
1453	-100

Figure 9(Input Procedure for Loading Information)

Support Information

There is not displacement possibility for some Dofs. Some degrees of freedom(Dof) were applied by loading .these information should be written in "restrain.txt". as Figure 10 The first column is Dof number that is supported.



1
5
11
21
33
51
71

Figure 10(Input Procedure for Support Information)

After running the program, displacement matrix (u) as the outputs is available through the program.

Example2: Asymmetric cracked plate with a hole

The capability of the proposed algorithm for structured mesh has been demonstrated in the previous examples, but this algorithm can also be generalized to unstructured mesh. An asymmetric cracked plate with a hole was selected. The plate was tensioned from two sides and meshed by unstructured quadrilateral elements as shown in Figure 11. Substructuring for each region is distinct.Because of General information and geometry described in the previous example,Only the information that is unique to this new example will be explained. The distinction of this example with the previous example is arbitrary substructuring and definition of different regions .

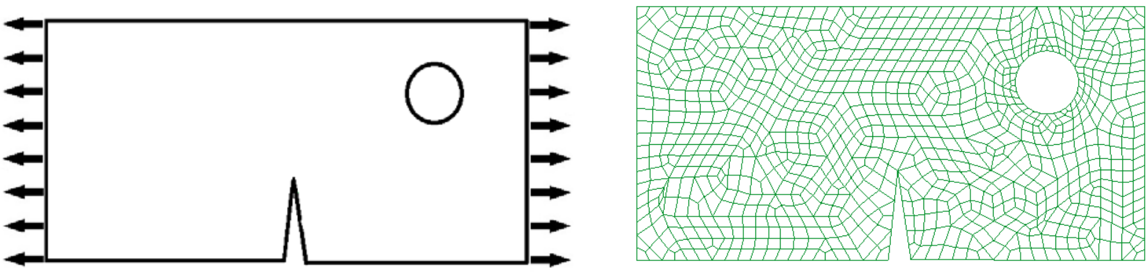


Figure 11(Geometry & Meshing of Example 2)

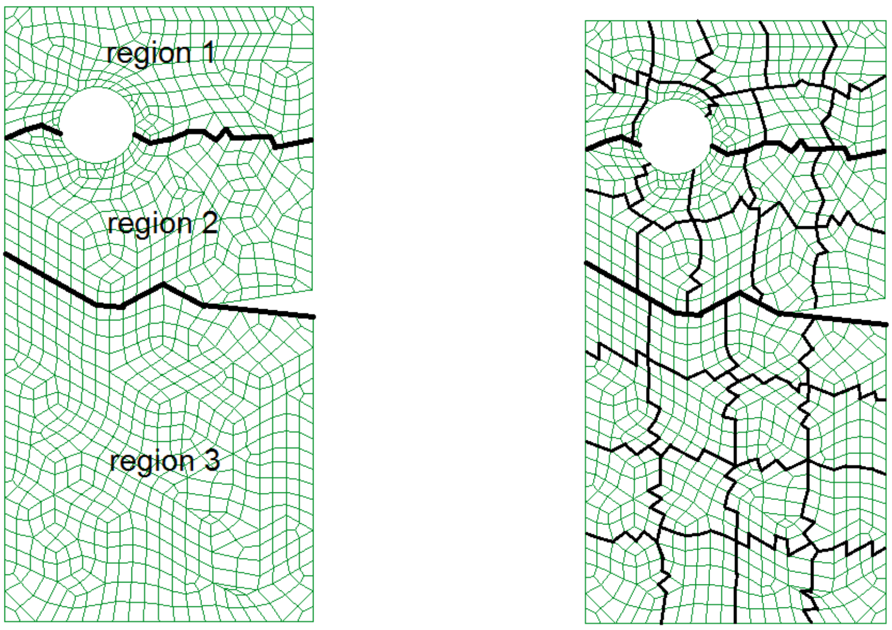


Figure 12(3 regions and subdivisions in each region)

There are 3 regions in the meshing and user should be create 3 folder with the name of region1,region2,region3. It should be noted that The folder name must be formed from the keyword "region" with the number of the region. In each folder, There must be a text files with the name of sub1, sub2, ... , sub(n).each of these text contains a one substructure informations. "n" is the number of substructures in the region. For example there are 10 substructures in the region1. Substructure informations means that element informations that included in the substructure .

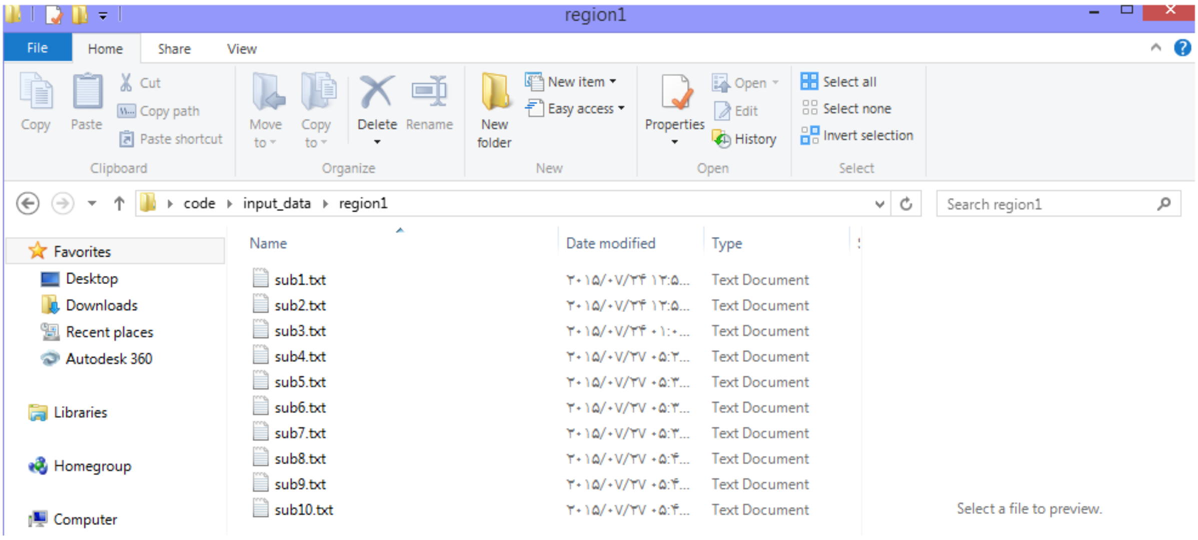


Figure 13(example of region folder)

For example, information of sub1 in region1 is shown in the Figure 14

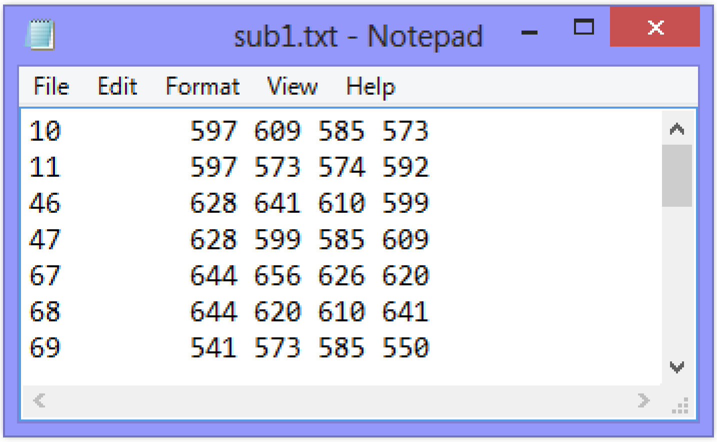


Figure 14(example of sub.txt)