ADAPT-Floor Pro

GUIDELINES FOR MESHING AND FINITE ELEMENT ANALYSIS

Dr. Bijan O. Aalami
Structural Engineer
Redwood City, California

Copyright 2001

Consulting Company Member

email: Support@AdaptSoft.com  website: http://www.AdaptSoft.com
1733 Woodside Road, Redwood City, California, 94061, USA,  Tel: (650) 306-2400  FAX: (650) 364-4678
LIST OF CONTENTS

1. DEFINITION OF TERMS .................................................................................................... 1

2. NOTES ON THE GENERATION OF STRUCTURAL MODEL .................................... 1
   2.1 MODELING OF STRUCTURAL COMPONENTS ............................................ 1
   2.2 MODELING OF SUPPORTS ................................................................. 5
   2.3 APPLIED LOADING .............................................................................. 6

3. FINITE ELEMENT DISCRETIZATION ........................................................................... 7
   3.1 ELEMENTS USED IN THE PROGRAM ............................................... 7
   3.2 STEPS IN DISCRETIZATION ............................................................... 8
       3.2.1 Imposed Nodes for Mesh Generation ......................................... 8
       3.2.2 Consolidation of Imposed Nodes ............................................... 11
       3.2.3 Maximum Mesh Size .................................................................. 21

4. PREPARATION OF THE STRUCTURAL MODEL FOR MESH GENERATION ............... 24
   4.1 FIRST STEPS IN PREPARATION FOR AUTOMATIC MESHING ............... 24
   4.2 MANUAL NODE SHIFT (NODE CONSOLIDATION) .................................. 27
   4.3 EXCLUSION OF LOCATIONS OF YOUR CHOICE FROM NODE SHIFT
       (NODE CONSOLIDATION) .................................................................... 28
   4.4 WALL INTERSECTIONS ........................................................................ 30

5 MESH GENERATION PREPROCESSING ..................................................................... 31
   5.1 RULE #1 – BREAK WALLS ACROSS DIFFERENT SLAB REGIONS .......... 31
   5.2 RULE #2 – BREAK WALLS ADJACENT TO MULTIPLE JAGGEDNESS
       OF SLAB BOUNDARY ......................................................................... 32
   5.3 RULE #3 – EXTEND OPENINGS TO SLAB EDGE ............................... 34
   5.4 RULE #4 – CONSOLIDATE ADJACENT OPENINGS ................................ 35
   5.5 RULE #5 – BREAK UP WALLS AND BEAMS THAT INTERSECT
       OUTSIDE A SLAB REGION .................................................................. 36
   5.6 RULE #6 – EITHER ELIMINATE SHORT WALL PROJECTIONS, OR USE
       MANUAL SHIFT NODE ....................................................................... 37

6. SUGGESTIONS FOR IMPROVED MESHING ............................................................. 38

7. MANUAL EDITING OF AUTOMATICALLY GENERATED MESH ................................. 38

8. AUTOMATIC MESHING USING EXCLUDER ............................................................. 45

9. MANUAL MESHING ................................................................................................. 48
   9.1 OVERVIEW AND QUADRILATERAL CELLS ........................................ 48
   9.2 GENERATION OF TRIANGULAR CELLS ............................................. 55
10. GLOSSARY OF TERMS .................................................................................................................. 59
10.1 NATURAL NODES .................................................................................................................... 59
10.2 ANALYSIS NODES .................................................................................................................. 59
10.3 OFFSET .................................................................................................................................... 61
FLOOR-Pro Module

GUIDELINES FOR MESHING AND FINITE ELEMENT ANALYSIS

This document provides the information necessary for a successful and expedient meshing of a structural model for analysis and design using ADAPT-Floor Pro. It explains the essential steps that need to be meticulously followed. All first time users of the program should thoroughly review this document and go through its tutorials.

1. DEFINITION OF TERMS

Several terms used in this writing are given specific meaning. Refer to the Glossary of Terms at the end of this writing for their definition. These terms are:

- Analysis node
- Frame element
- Natural node
- Node shift
- Offset
- Reference plane
- Shell element
- Tendon segment

2. NOTES ON THE GENERATION OF STRUCTURAL MODEL

2.1 Modeling of Structural Components

For the gravity design, each floor level is modeled together with the supports immediately below and above it. The supports can be columns, walls, springs, points or lines. Soil supports are treated separately. The geometry of a floor is defined with respect to a horizontal plane. The distance from the top of different parts of the floor to this horizontal plane determine the changes in the elevation of the floor, if any. This horizontal plane is referred to as the “Current Plane,” It is also referred to as “Current Reference Plane.” Figure 2.1-1 shows an example of a current reference plane and the default positioning of several structural components with respect to this reference plane. For an example, note that the top of the beam and the drop cap are lined up with the current reference plane. When a structural component, such as a slab region does not lie at the location shown, it has to be moved to its correct position. This is done using the “offset” feature of the program. The offset feature is described later in this document.
The supports below the floor system are defined with reference to the current plane and bottom plane (Fig. 2.1-2). Similarly the supports above the floor system, such as columns, walls, ramps, are defined with respect to the current plane and the upper plane. The upper and lower reference planes are used by the program to determine the height of each of the supports. For example, the length of an upper column is defined by the position of its top and bottom ends. The position of the bottom end is defined with respect to the current plane, and the position of the top end with respect to the upper reference plane. The default of the program is that each wall or column support extends between the current plane and either the upper or lower plane. For components that do not extend over this length, such as the short upper column shown in Fig. 2.1-2, the offsets (Z distance) from the associated reference planes are used to adjust the ends of the member.
The default position of the structural components with respect to the current plane is shown in Fig. 2.1-1. But, once created, a structural component can be automatically adjusted by the program to its natural position. For example, the top of column below a slab will be automatically moved to the soffit of slab. Hence, there will be no overlap or interference between the top of a column and slab region.

Figure 2.1-3 illustrates the concept. The partial elevation of a floor with a raised region and slabs at different levels on each side of the raised region is shown in Fig 2.1-3a. In part (b), the slab region marked “F” is assumed to be at the level of the reference plane. The position of the remainder components of the floor system are defined by their Z-offset. For example, region “D” has a negative offset ($Z_D$), while region “E” has a positive offset ($Z_E$). Once the level of the slab regions is established, the program’s utilities will automatically adjust the position of the columns A, B and C to extend to either the soffit or the top of the relating slab regions.
ADAPT GUIDELINES FOR MESHING FLOOR-Pro

FIGURE 2.1-3

(a) ELEVATION

(b) OFFSET IDENTIFICATION

ADJUSTMENT OF LEVELS USING OFFSET

FIGURE 2.1-3
2.2 Modeling of Supports

Supports, as illustrated in Fig. 2.2-1 are positioned at the reference line. They have to be shifted to their final location prior to the analysis.

Figure 2.2.2 shows a condition where the position of supports affects the solution.
2.3 Applied Loading

Applied loads are positioned at the reference plane. They should be moved to their correct position, if they do not act at the reference plane.
3. FINITE ELEMENT DISCRETIZATION

3.1 Elements Used in the Program

The structural system of the floor is modeled using “Structural Components,” such as slab regions, walls and columns. The structural model, together with the support conditions, such as the type of fixity at the bottom of the columns, and the loading will be solved (analyzed) by the software. The solution (analysis results) yields displacements, stresses and actions (moments, shears, axial forces) of the floor system under the given loading.

ADAPT uses the Finite Element Method (FEM) to solve the structural model. In using FEM, the structural model will be represented by “elements” that are joined to one another at common nodes. The break down of the structural model into finite elements is referred to as “discretization.” In the general case, the discretization - also referred to as “meshing” - of the structure is done automatically by ADAPT. However, some knowledge of the way ADAPT does the discretization is of great help in achieving a good and expedient solution. This document explains the concept and the information necessary for a successful discretization “meshing” of the structural model.

ADAPT uses several different types of elements in its discretization. These are:

- Flat quadrilateral or triangular shell elements. ADAPT’s automatic mesher creates quadrilateral elements only. However, the user has the
option to create triangular elements when using the manual meshing or editing options of the software.

- Frame elements: prismatic frames with one node at each end having six degrees of freedom
- Spring elements with six degrees of freedom at each end
- Truss elements with ability to resist axial load only
- Soil elements for modeling Winkler foundation

The selection of elements for the various structural components is as follows:

<table>
<thead>
<tr>
<th>Element Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shell Elements</td>
<td>slab regions, drop caps, drop panels, walls, ramps, slab bands</td>
</tr>
<tr>
<td>Frame Elements</td>
<td>beams, columns</td>
</tr>
<tr>
<td>Truss Elements</td>
<td>prestressing and longitudinal rebar</td>
</tr>
<tr>
<td>Soil</td>
<td>spring with axial stiffness only</td>
</tr>
<tr>
<td>Miscellaneous</td>
<td>springs with six degrees of stiffness in direction of global axes</td>
</tr>
</tbody>
</table>

### 3.2 Steps in Discretization

The steps in the breakdown (discretization) of a structural model into “elements” for the finite element analysis are as follows:

- Preparation of the structural model for mesh generation
- Mesh generation; mesh editing
- Conversion of generated mesh into finite elements

Once the finite elements are generated, the program can solve the structure for displacement and actions. Each of above three steps will be described in detail next.

### 3.2.1 Imposed Nodes for Mesh Generation

The structural model is made up of structural components, such as beams, columns, slab regions, drop caps and more. Each of these components is meshed (subdivided into elements) on its own with the provision that there will be common nodes at the intersection, or connection of each structural component with the others.

At the initiation, each of the structural components is positioned with respect to the current reference plane as shown in **Fig. 3.2.1-1**. Note that the analysis node of the structural component at the reference plane is placed on the reference plane immediately above the associated natural node. Refer to the glossary of terms for additional details on analysis and natural nodes.
Prior to the generation of automatic meshing, the program automatically imposes a number of nodes at selected locations. These are intended to delineate the structural components for the finite element meshing. The imposed nodes are at locations such as:

(i) The end of walls, beams and columns;
(ii) at intersection of structural components, such as a beam and a wall; and
(iii) at the corners of slab regions, openings, drop cap and similar condition.

Similarly, there will be imposed lines along which the automatic mesher will be generating nodes. Examples of these conditions are:

(i) Along the edge of slab regions;
(ii) along the edges of openings;
(iii) along the length of beams, columns, and intersection of a wall with slab, and similar condition.

Once the location of the imposed nodes are determined, the program will automatically mesh the remainder of the structural model using a user defined maximum mesh size.

**Figure 3.2.1-2** is an example of imposed nodes for automatic meshing. It shows partial plan of a slab supported on two adjacent walls. The imposed nodes are at the ends of the walls, and in the slab at locations associated with
end of walls. In part (b) of the figure the end of the wall points A and B signify imposed nodes. Other nodes will be generated between A and B along the imposed line AB. The number of lines generated by the program along the line AB depend on the user’s preferred mesh size.

An example of imposed nodes, imposed lines and automatic meshing is illustrated in Fig. 3.2.1-3. Note that the meshing generated automatically by the program includes the imposed nodes and has placed new nodes along the imposed lines.
3.2.2 Consolidation of Imposed Nodes

Consider the wall supported slab shown in Fig. 3.2.2-1. The gap between the two walls, as shown in part (b) of the figure is about the same as the slab thickness. For a gap up to approximately four times the slab thickness, the behavior of the region spanning the gap is significantly different from the assumption of “plane sections remain plain,” which is the basis of ADAPT program and essentially all other linear elastic finite element analysis schemes. Such regions are referred to as discontinuity (D regions) in the literature (Fig. 3.2.2-2b). The smaller the gap, the closer is the response of the region to a solid mass. Normally, imposed nodes are placed at the corners of the walls together with imposed lines as shown in parts (c) and (d) of the figure.
It is expedient and safe from a structural design standpoint, if the natural nodes of the members terminating at such regions are consolidated into a single analysis node, such as node A shown in part (c) of the figure. Node A will be linked to the natural nodes 1, 2 and 3 by way of three offsets. The consequence of this modeling is that the three points 1, 2 and 3 will be acting as if rigidly connected together. The conditions of equilibrium with the applied loading, necessary for the safety of the structure is maintained. The approximation affects the distribution of stresses within the D region. But, the analy-
sis will conclude with each of the three natural nodes having its own displacement and forces.

FIGURE 3.2.2-2

Structural design is based on the magnitude of the actions (moments, shears, axial loading) at selected design sections. In ADAPT formulation, contrary to traditional FEM [Aalami, Dec. 2001] the design actions at a section are not derived from the distribution of internal stresses. The design actions are based on the forces at the nodes adjacent to the section. The consolidation of nodes
at a “D” region leads to an expedient and safe design. The regions are designed to satisfy the equilibrium forces and detailed for ductility.

In Fig. 3.2.2-2(c), the position of node 1 with respect to its analysis node is shifted both horizontally and vertically. The horizontal shift is referred to as “Node Shift.” The node shift is generally done automatically by the program. User also has the option to shift nodes manually and consolidate several natural nodes of a D region into a single analysis node, or to suppress node shift altogether. The automatic shift node of the program takes place when two or more “imposed” natural nodes fall closer to one another than a user defined distance. The distance is generally two to four times the slab thickness.

One of the great advantages of the node consolidation is the substantial reduction in the number of nodes necessary to complete a design. Clusters of closely spaced imposed nodes result in a dense meshing around them and subsequently a greater demand on the computational resources with no significant increase in the accuracy of the design results.

As an example, consider the partial plan of slab shown in Fig. 3.2.2-3(a), showing a wall, beam and column closely spaced to one another. The natural imposed nodes of each is shown in part (b) of the figure. The region bound by the closely spaced components acts essentially as a chunk of solid and rigid concrete. The shift node option of the program would represent the region by a single analysis node as shown in part (c) of the figure. It is important to note that while the three natural nodes shown are consolidated into a single analysis node, the program will automatically determine and report the displacement and actions at each of the natural nodes separately. This determination is based on satisfying the equilibrium of the forces within the region common to the natural nodes consolidated and considering the displacement of the region as a rigid chunk of concrete.
(a) PARTIAL PLAN

(b) IMPOSED NATURAL NODES

(c) CONSOLIDATED ANALYSIS NODE

CONSOLIDATION OF IMPOSED NODES

FIGURE 3.2.2-3
A common practical scenario of node consolidation is the condition of walls terminating at the slab edge. **Figure 3.2.2-4** illustrates the traditional FEM modeling (part b) and the method developed by ADAPT (part c).

**Figure 3.2.2-4**

The common cases of node consolidation done automatically by the program are listed in **Fig. 3.2.2-5**
FIGURE 3.2.2-5a

MODELING OF STEPS IN SLAB AND BEAM SLAB CONNECTION
<table>
<thead>
<tr>
<th>ACTUAL</th>
<th>MODEL</th>
</tr>
</thead>
<tbody>
<tr>
<td>(iv)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SLAB</td>
</tr>
<tr>
<td>SECTION</td>
<td></td>
</tr>
<tr>
<td>(v)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>BEAM</td>
</tr>
<tr>
<td>SECTION</td>
<td></td>
</tr>
<tr>
<td>(vi)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SLAB</td>
</tr>
<tr>
<td></td>
<td>SLOPE</td>
</tr>
<tr>
<td>SECTION</td>
<td></td>
</tr>
<tr>
<td>(vii)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>BEAM</td>
</tr>
<tr>
<td>PLAN</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>SECTION A-A</td>
<td></td>
</tr>
</tbody>
</table>

GUIDELINES FOR MODELING OF STEPS IN MEMBER AND BEAM SLAB CONNECTIONS

FIGURE 3.2.2-5b
FIGURE 3.2.2-5c

COLUMN MODELING
Figures 3.2.2-6 and 3.2.2-7 show other examples of node consolidation and offset.
3.2.3 Maximum Mesh Size

The automatic meshing is attempted by the program using a maximum mesh size defined by the user. Generally, 8 to 10 divisions for the typical span of a floor system give a good solution for design. If complexity in geometry of the structural model makes it impractical to mesh with the user suggested size, the program automatically reduces the mesh size and makes a new attempt. The reduction of maximum mesh size followed by a new attempt at meshing is continued through many iterations until either a successful meshing is
achieved, or the number of iterations allowed are exhausted. If the automatic meshing is successful, the user has the option to view/edit the meshing selected. Otherwise the solution continues to determine the design values.

If meshing is not successful, several options are available.

- Re-attempt automatic meshing, but specify a smaller “maximum mesh size.” The value to be specified should be less than 20% of the previous attempt, since the program automatically reduces the suggested size in each of its iterations until it reaches a size about 20% of the user suggested dimension. For example, if the first user suggestion was 2m, the second suggestion should be 0.4m or less.

- Ask the program to display where it is encountering difficulty in meshing. The program will encircle and display the location(s). A visual evaluation of the location reveals the preparatory steps necessary for the meshing. Invariably, these are locations where cluster of closely spaced imposed nodes, such as intersection of several beams at an uncommon location makes it difficult for automatic meshing. In this case the user should rectify the location indicated by the program by manually consolidating two or more of the imposed nodes. This is referred to as “manual shift node” operation. The concept of shift node was described earlier and its detailed procedure is explained later in the following

![Diagram](image-url)

(a) Selection of a large “Maximum Mesh Size”

![Diagram](image-url)

(b) Selection of a small “Maximum Mesh Size”

**FIGURE 3.2.3-1 EFFECT OF “MAXIMUM MESH SIZE” SELECTION**
The effect of the maximum mesh size selection on the outcome of the program’s automatic meshing is illustrated in the following figures which show the plan of a slab with an opening.

The maximum mesh size selection is made in the dialog box shown below when invoking the automatic mesh generation button.

![Maximum Mesh Size Setting Dialog Box](image)

**FIGURE 3.2.3-2 MAXIMUM MESH SIZE SETTING DIALOG BOX**

The final mesh size selected by the program is governed by the location on the floor system where closely spaced imposed nodes dictate a dense meshing. The following example illustrates the point.

*Figure 3.2.3-3(a)* shows a slab region with corner columns. For analysis, it is advantageous to consolidate the analysis node of this column with that of the slab corner, in other words shift the column node to the corner of slab. If the column node is not shifted to the slab corner, the program will impose a node at the column centerline and mesh the slab region accordingly (part b of the figure). The outcome is generally a denser mesh, since the distance between the column centerline and the slab edge must be represented by at least one slab element. However, if the column centerline be shifted to the slab corner (part c of the figure) the imposed node will be at the slab corner. The ensuing mesh generation will be independent of the column’s natural node. The outcome is a larger mesh size (fewer elements) as shown in part (c) of the figure. It is emphasized that the location of the corner column is unchanged, but for purposes of analysis, its natural node is advantageously shifted to the slab corner.

Edge columns are treated in a similar manner. Their natural node is shifted to the slab edge, where the program will automatically impose an analysis node.
4. PREPARATION OF THE STRUCTURAL MODEL FOR MESH GENERATION

From the foregoing, it becomes apparent that prior to meshing the user should examine the structural model for possible consolidation of closely spaced imposed nodes and measures that lead to an optimum and well balanced meshing. This section provides the guideline for the steps to follow.

4.1 First Steps in Preparation for Automatic Meshing

- Inspect the plan and identify the possible critical span. Select a mesh size to have this span represented by 8 to 10 divisions. Shorter spans are likely to end up with a smaller number of subdivisions. In most designs, this is not a critical issue.
- Consider a distance for node shift equal to three times the average slab thickness. A distance between two to four times gives reasonable results.
- Select automatic meshing. Unselect “Solve after meshing.”
- Go to “Options” and set the “Minimum Distance” of node shift to value you selected (Fig. 4.1-1).
Start the automatic meshing.
  - If the program concludes with “successful meshing,” view the mesh using either the “View Meshing” item from the FEM pull down menu, or from the “Select/Set View Items” tool button.

**Figure 4.1-2** shows plan of a floor system with closely spaced columns. When using the “Sift nodes automatically” option, the imposed top node of the closely spaced columns will be consolidated into a single analysis node. The resulting meshing is shown in **Fig. 4.1-3**. A close up of the three-column cluster (**Fig. 4.1-4a**) indicates a node at center of one of the columns. You can verify the consolidation of column slab connection of the three columns into one, by displaying the column symbol in Select/Set View Items” tool button.

**Figure 4.1-4b** displays the consolidation of the column-slab nodes of the two left columns with the one on the right. It is emphasized again that each column is meshed independently and is subjected to the action at its connection to the slab (**Fig. 4.1-5**).
FIGURE 4.1-3 MESHING OF THE STRUCTURE USING NODE SHIFT

FIGURE 4.1-4 CONSOLIDATION OF THREE CLOSELY SPACED COLUMN TOP NODES INTO ONE (NODE SHIFT)

(a) Frame Element Symbol Not Shown  (b) Frame Element Symbol Shown
If the program does not conclude with successful meshing, respond yes to the screen question of whether you want to see the problem areas.

4.2 Manual Node Shift (Node Consolidation)

To simplify the meshing, it may become necessary to shift the node of a column, a wall, or a beam manually. The manual shift node is accessed through the FEM pull down submenu “Shift Nodes.” The following options are available:

- Shift Nodes Automatically
- Shift Nodes Manually
- Cancel Node Shift
- Detect Potential Problems

Shift node option applies to columns, beams, walls, point supports, and line supports. Once a node is shifted manually, it will remain at its imposed position until you activate the “Cancel Node Shift.” The following illustrates the procedure.

**Figure 4.2-1a** shows a floor slab with two columns and a beam. The node of column A and the nodes of the beam are shifted manually to the locations A for the column and B,C for the beam (**Fig. 4.2-1b**). The new locations are displayed with the symbol of the column and the beam. The analysis nodes generated in meshing for the beam
will be along line BC. Likewise, the node of column at the top left of the slab will be at point A. The analysis node of column D was not shifted. Hence, a node is generated at its center (part (c) of the figure).

![Diagram of node shifts and meshing](image)

**FIGURE 4.2-1 MANUAL NODE SHIFTS AND THE RESULTING MESHING**

4.3 Exclusion of Locations of Your Choice from Node Shift (Node Consolidation)

You can selectively freeze one or more components from node shift, while allowing the analysis nodes of the remainder of the eligible components to be automatically, or manually shifted. Consider the floor plan in **Fig. 4.3-1a**. Column A is well within the interior of the slab and is not a natural candidate for automatic node shift. The remainder of the columns B, C and D, along with the two beams E and F are close enough to the slab edge to be subject to node shift. Let us assume that you do not want the node of column C and the nodes of beam F to be shifted. The following is the procedure:

- Double click on column C to open up its property dialog window (**Fig. 4.3-2**).
- Select the Finite Element tab
- Select the check box for “Do not shift nodes.” This will lock the position of the analysis node of this column at its natural position.
(centroid of the column). The two data fields to the top right of this check box refer to the offset values of the node shift in the X- and Y-directions.

- Likewise, double click on beam F to open its property box and select the “Do not shift nodes” check box in its Finite Elements tab.

**Figure 4.3-1b** shows the position of the shifted analysis nodes of the columns and the beam which were either not frozen, or were not a candidate (column A). The outcome of meshing is shown in part (c) of the figure. Observe that there are nodes on column C and along beam E, the two structural components with frozen node shifts.

The mesh layout resulting from this operation is shown in **Fig. 4.3-1c**. Note that there are no nodes

![Diagram](image1)

**FIGURE 4.3-1 COMPONENTS WITH NODE PERMITTED AND LOCKED NODE SHIFT**
4.4 Wall Intersections

There are several options to model the intersection of walls. These are illustrated in Fig. 4.4.1-1. At intersection “A” (part c of the figure), the walls intersect. Likewise, at “B” wall 7 is modeled to overlap with wall 6. Wall 9 butts to wall 8 at intersection “D.” Using node shift option, the resulting meshing is shown in part (c) of the figure. Note that in all instances, the intersection is consolidated into a common node. The difference between the different modeling schemes lies in the meshing and design of the wall itself. For example for calculation of building weight and wall design, the length of wall 10 is calculated to the face of wall 8. But the length of wall 5 is calculated to the outside face of wall 6, that is to say, wall 5 is assumed to extend over the entire width of the slab.
Before attempting the automatic mesh generation of your floor system, it is necessary to review the structural model as described below and make the necessary adjustments for a successful and balanced meshing. The steps to follow are described as rules.

5.1 Rule #1 – Break Walls Across Different Slab Regions

Where walls extend over more than one slab region, such as shown in Fig. 5.1-1b, they must be broken into several parts, each covering one of the slab regions. The wall in part (b) of the figure is continuous, but is broken into three parts. When the “Establish Compatibility” tool of the program is invoked, the program automatically...
ADAPT GUIDELINES FOR MESHING FLOOR-Pro

adjusts the top of each wall to be in line with the soffit of respective slab region. The separation of the walls shown in part (b) of the figure does not affect the analysis of the structure. In the analysis, the walls that touch one another, such as this case, will be considered contiguous and will be treated as one.

![Figure 5.1-1 Walls Extending Across Slab Regions](image)

(a) (b)

5.2 Rule #2 – Break Walls Adjacent to Multiple Jaggedness of Slab Boundary

Where a wall is close to a multiple discontinuity (jaggedness) at slab edge or opening, as illustrated in Fig. 5.2-1, it may be necessary to break the wall up into sections. The break up is only necessary if there is more than one protrusion or cut out along the slab edge, and in addition if you invoke the “node shift” option of the program. When the node shift option is invoked, the program attempts to shift the analysis axis of the wall onto the slab edge. If there slab edge is discontinuous it is not possible to line up the wall axis with more than one section of the slab edge. For this reason, the wall has to be broken down into sections as shown in part (b) of the figure. The number of sections should correspond to the pieces at slab edge adjacent to the wall.

It is reiterated that the breakdown of the wall is not necessary, if shift node option is not used. Also, for the analysis the program will automatically considers the subdivided pieces of the wall as whole. In other words, the breakdown of the wall into segments does not impact the solution.
FIGURE 5.2-1 BREAKUP OF WALLS NEXT TO JAGGED SLAB EDGE

For example in Fig. 5.2-2 wall A need not be subdivided regardless of whether or not shift node utility is used, since its axis can be uniquely shifted to the slab edge next to it. Wall be adjacent to the two openings, however, needs to be broken down if shift node is used. This wall is possibly closer to two opening edges than the shift node default value. It is not possible for it to line up with both opening edges. Hence, it has to be broken down. It is best to break it down into a minimum of two pieces.

A variation of the above conditions is shown in Fig. 5.2-3, where a wall is adjacent to a slab edge and at the same time its extension is close to an opening. Again, it will generally not be possible to line up a parallel to the axis of the wall with both the slab edge and the edge of the opening. If automatic shift node is to be used, you need to break the wall into segments.

FIGURE 5.2-2 WALL "A" NEED NOT BE BROKEN UP, BUT WALL "B" MAY HAVE TO
5.3 Rule #3 – Extend Openings to Slab Edge

Where openings are next to slab edge, such as shown in three different cases in Fig. 5.3-1, you need to extend the opening to the slab edge. The extension treatment is necessary if the width of the slab band between the opening and the slab edge is too narrow to be meshed. Or, the slab band is not a significant in the load carrying member of the floor system.

Part (b) of the figure shows the treatment in each of the three cases. If the slab band between the opening and slab edge is not supported, the slab boundary is moved around the opening. When a wall or a beam separates the opening from the slab edge, the opening is extended to the slab edge with no modifications made to either the wall or the beam. In the analysis the wall and the beam forming the outer boundary of the opening are automatically meshed and their contribution in resisting the applied loading is accounted for in full.
5.4 Rule #4 – Consolidate Adjacent Openings

When two or more openings are drawn next to one another, such as the two openings at the left of Fig. 5.4-1, they should be combined into one opening as shown in part (c) of the figure. If left as is, it is possible that the automatic meshing could treat it successfully (part b of the figure), but this is not always guaranteed in congested areas.

Likewise, when two openings are drawn one on each side of a wall or a beam, it the openings should be combined as shown in part (c) of the figure. When combined, the program correctly accounts for the continuity and load carrying features of the wall or beam that passes through the opening.

FIGURE 5.3-1 TREATMENT OF OPENINGS NEXT TO SLAB EDGE
5.5 Rule #5 – Break Up Walls and Beams That Intersect Outside a Slab Region

Figure 5.5-1 shows a slab region with three walls A, B and C. Wall A does not intersect with another wall outside the slab area. This wall needs not special treatment. Walls B and C, however, intersect at a point outside the slab region. You need to break these walls into segments. The portion of each wall that falls outside the slab region must be entered as a new wall. This is a modeling requirement. From analysis standpoint, the program will automatically consider each of the walls contiguous.
5.6 Rule #6 – Either Eliminate Short Wall Projections, or Use Manual Shift Node

Where short walls project from longer walls they may need to be treated as follows:

- If the wall is very short, such as projection B in Fig. 5.6-1, and its presence is primarily for architectural reasons, select the wall and change in its property window the option “disregarded.” This will retain the wall for completeness of the architectural features and authentic preparation of structural drawings, but will not consider it as a load bearing member. In other words, by selecting “disregard” you convert a load bearing member to a “non-load bearing” member.

- If the wall projection is medium long (ration of projection length to thickness 4 or less), such as projections D, E and F, you need to manually set the finite element nodes located at their ends farther apart. This will avoid localized congested meshing. The manually separation of the end nodes is done using the “manual node shift” option of the program. Figure 5.6-1(b) shows the shifted axis of the wall projections D, E and F. Note that one end of the wall projection is snapped to backbone long wall, and the other end is extended out to a distance equal or greater than the optimum mesh size you select.

- For wall projections with aspect ratio greater than four, generally no preprocessing treatment is necessary.

![Figures 5.6-1(a) and 5.6-1(b)](image)

**FIGURE 5.6-1 TREATMENT OF SHORT WALL PROJECTIONS**
6. **SUGGESTIONS FOR IMPROVED MESHING**

The following recommendation can help improve the meshing, in particular where closely spaced multiple constraints result in a dense meshing.

Extend the Boundaries of Openings to Centerline of Adjacent Walls

![Figure 6.1-1 Opening Extended to Centerline of Walls](image)

**FIGURE 6.1-1 OPENING EXTENDED TO CENTERLINE OF WALLS**

7. **MANUAL EDITING OF AUTOMATICALLY GENERATED MESH**

This section describes how you can edit a mesh created by the automatic mesh generation of the program. Manual mesh generation or editing becomes necessary under at least two conditions.

First, in some instances the automatic meshing creates a pattern that at selected locations in the structure it is unnecessarily too dense. The dense locations can be manually edited to a more balanced mesh layout. This leads to faster program operation.

Second, at some locations the mesh generated by the program may have an unfavorable shape or aspect ratio. You may wish to improve the geometry at these locations. The program gives solutions acceptable for design of aspects ratios well beyond 50. For quadrilateral shell elements this is the ratio of the element length to the element width. For triangular elements this is the ratio of the triangle base to its height. In such instances, the program prompts a message on the computer screen and identifies the mesh you need to edit.

The following walks you through the steps of manual mesh editing by way of an example. Consider the floor slab shown in Fig. 7.1. Let us edit the meshing around the upper right hand column to a lesser density and more balanced pattern.
FIGURE 7.1 FLOOR PLAN SHOWING FINITE ELEMENT MESHING

The enlarged view of this region on the computer screen is given in Fig. 7.2. The quadrilaterals and/or triangles intended for editing are first selected (Fig. 7.3). The selected items are identified on the screen by a change in color. The selected items are then deleted (Fig. 7.4).

FIGURE 7.2 COMPUTER SCREEN ZOOMED AT LOCATION FOR EDITING
FIGURE 7.3 QUADRILATERALS INTENDED FOR EDITING ARE SELECTED

FIGURE 7.4 ITEMS SELECTED FOR EDITING ARE DELETED
From the FEM pull down menu select “Manual Meshing” followed by selection of “Create Mesh for Slab Elements” Slab with Shell Elements” (Figs. 7.5 and 7.6).

Set the number of cells to be created to 1 as shown in Fig. 7.7. Press OK to close this dialog window and start drawing quadrilateral or triangular cells to fill in the region to be meshed. Complete each cell first before going to the next. In drawing the cells you need to use the appropriate snapping tool from the “Snap Toolbar” shown in Fig. 7.8. Use “Snap to Intersection” (fourth button from the left) wherever possible. This will ensure the connectivity of adjacent cells at the nodes.
Fig. 7.9 shows the first two cells created. Note that column center is selected as one of the nodes. The completion of the work is shown in Fig. 7.10.

A faster method to fill in a specific region in a mesh is to generate a group of cells that cover the area of the region and then snap the perimeter nodes of the new mesh to the boundary of the enveloping mesh. Before getting started, you will need to approximate how many cells
are necessary to fill the region. To create this cell mesh, you will use the same tools as in the previous example. From the FEM pull down menu select “Manual Meshing” followed by selection of “Draw a Slab with Shell Elements” (Figs. 7.5 and 7.6). However in the “Shell Elements – User Defined” dialog box, you will define several cells at a time instead of one-by-one. To fill this region a mesh of 3 X 2 will be used. Therefore you must define the mesh as shown in Fig. 7.11

![Shell Elements - User Defined Dialog Box](image)

**FIGURE 7.11 USER DEFINED MESH DIALOG BOX**

Once the mesh has been defined and click “OK”, the program will prompt you to enter the four boundary points of the 3 X 2 mesh. It is advantageous to position the four points so that the new mesh’s boundary is very close to the boundary of the enveloping mesh (Fig. 7.12). If the new mesh is positioned well, then there will be less work in adjusting the shells to get an even mesh.

![View of 3 X 2 Mesh](image)

**FIGURE 7.12 VIEW OF 3 X 2 MESH**

Use the diagrams in Fig. 7.13 as a guideline in positioning the cell mesh. Note that to reposition a common node shared by two or more cells, you must select all meet that meet at that common point. To do so, click on the first cell and then hold down the “Ctrl” key to add more cells to the selection. If you have difficulty selecting a cell, you may find the “tab” key function helpful in selecting overlapping cells. After the cell mesh and the enveloping mesh
are joined as one, you must position a common node over the columns natural node as shown in Fig.7.13 (d). The mesh is complete.

FIGURE 7.13 NODE ADJUSTMENT PROCESS
8. AUTOMATIC MESHING USING EXCLUDER

**Figure 8-1** LAYOUT IN PLAN VIEW

**Figure 8-1** shows a slab with a wall, columns and an opening. The proximity of the wall to the cut out corner and a column to an opening is likely to cause a dense mesh within the region. If meshing is attempted and is successful, a layout as shown in **Fig. 8-2** with 541 elements will result.

It may be desirable to use an alternative option to this layout for two reasons. First, the mesher may not be successful if some of the structural components are spaced too closely. When the mesher is not successful, circles will appear around the regions that create problems (**Fig. 8-3**). In many cases, the problem is that the components are closer in distance than the given mesh size. Secondly, if the user inputs a small mesh size, the mesher may create a mesh that is too dense and not justifiable. To resolve this dilemma, the user can exclude these regions and manually mesh them.

**Figure 8-2** MESHING WITHOUT EXCLUDER (541 ELEMENTS)
FIGURE 8-3 PROBLEM REGIONS INDICATED BY CIRCLES

To exclude a slab region, click on the Excluder tool located in the “FEM” menu under “Mesh Manually”. A message will appear on the screen prompting you to draw a polygon around the region to be excluded. For the program to exclude the region successfully, there are four rules that need to be followed.

Rule #1: The first point of the excluder shall fall within the slab region.
Rule #2: The excluder cannot cross outside the boundary of a slab in more than two locations.
Rule #3: The area bounded by the excluder shall be convex. That is to say, none of the internal angles of an excluder shall exceed 180 degrees.
Rule #4: Excluder cannot intersect an opening, but can fully contain one or more openings.

Using the rules stated above, two excluders are drawn around the two problem areas as demonstrated in Fig. 8-4. The remainder of the slab region is now ready for Automatic Mesh Generation at a recommended mesh size of 3ft. The resulting mesh (Fig. 8-5) will have a more course and even grid. The components located inside the excluder will be treated according to the following rules.

Rule #1: Automatic shift node does not apply within the excluded area
Rule #2: Manual shift node is permissible within the excluded area but shall not be permitted to be shifted across or on the excluder boundary.
Rule #3: If automatic meshing is used the following is done:
The structural components that fall entirely within the excluded region will be meshed except the slab region.
If manual meshing is attempted within an excluded region, the rules and procedure of manual meshing apply.
If a solution is attempted, either following a manual meshing, or automatic meshing without filling the excluded regions with meshing. The program will not solve the problem at all.

FIGURE 8-4  TWO EXCLUDERS AROUND PROBLEM AREAS

FIGURE 8-5  AUTOMATIC MESH GENERATION AROUND EXCLUDED REGIONS

The regions excluded are now ready to be meshed manually. To mesh manually refer to the documentation called *Manual Mesh Editing*. 
9. MANUAL MESHING

9.1 Overview and Quadrilateral Cells

The following describes the procedure for manual meshing of ADAPT-Floor Pro and ADAPT-Floor RC. Manual meshing is one of the three options of meshing a structural model prior to using finite elements for its analysis. The three options are:

- Fully automatic meshing
- Automatic meshing in combination with “Excluder” and partial manual meshing
- Manual meshing

The manual meshing is limited to the discretization of slab regions, including drop caps and drop panels. Meshing of other components, such as beams, columns, and walls are carried out automatically without user interaction. Point supports, however, if directly on a slab region are included in the manual meshing. Supports of other components, such as support at the far ends of columns and walls are handled by the program automatically.

The “Node Shift” feature of the software, described in the guidelines for meshing, is also available for manual meshing. If required, the node shift option has to be exercised before manual meshing is started.

Once manual meshing is attempted, the program performs a “validation check” for the structural model to be meshed. The purpose of this validation check is to detect obvious errors, such as a column not having been attached to a slab. The validation also corrects, or improves a number of shortcomings in the structural model, such as merging points that are too close to one another, or multiple points that are mistakenly entered by the user.

Manual meshing is described herein by way of an example. Figure 9-1 shows the plan of a floor with an opening, three columns and a beam.

Manual meshing starts by invoking its icon from the FEM (Finite Element Method) pull down menu. The program performs a validation for the structural model in the background. If the program detects a problem, or shortcoming, it will display its finding on the computer screen either as an “error,” or a “warning.” The errors must be corrected by the user, before manual meshing can proceed. The warnings, however, can be ignored, if these are acceptable by the user.

After the structural validation, the program converts the floor system to a simple drawing that shows only the outline of the mesh cells. The outline of the mesh cells for the floor system of Fig. 9-1 is shown in Fig. 9-2.
The columns are shown by points in the outline. Beams, walls, opening and slab boundaries are shown by lines, each with its own color. The manual meshing should conclude with a “node” on each of the points on the outline plan (Fig. 9-2), and cell boundaries on each of the lines. Further, the cells should cover the entire floor area, except the openings.

At the next step, the program prompts a dialog box for cell divisions in each direction of a mesh to be created. This dialog box is shown in Fig. 9-3a for the current example. A complex floor plan is generally covered by several mesh sets. For the current example, one mesh set will be adequate. As shown in Fig. 9-3a, 10 and 8 divisions are selected.

After pressing “OK” on the cell divisions dialog box, the program asks the user to select the four corners of the mesh to be generated. This is done through prompts on
the command line at the bottom of the computer screen. At this stage, it is advanta-
geous to use the snap tools of the program and match the corners/sides of the mesh to
those of the points/lines on the mesh outline. In the current example the “snap to
intersection” tool is used. As it is illustrated in Figs. 9-3b and 9-3c, the corners of the
slab region are selected. Once the last corner at the bottom left is clicked, the mesh
shown in Fig. 9-3d appears.

![Dialog Box for Number of Cells in the Mesh](image1)

![Selection of First Point at Top Left](image2)

![Selection of the Second Point at Top Right](image3)

![Generated Mesh Over the Slab Region](image4)

**FIGURE 9-3 STEPS IN GENERATION OF A MESH REGION**

**Figures 9-4 through 9-9** illustrate the editing of the mesh to comply with the mesh
cell outline shown in Fig. 9-2. The editing consists of the following steps.

- Select the common node of the cells next to column points, drag them and
  snap them to the column points (Figs. 9-4 through 9-6). In this operation, it is
  important to select all the cells that share the common node to be shifted to the
column.

- Select the common nodes of the cells that are close to opening corners and
  using “snap to intersection” snap them to the adjacent opening corner.
  Follow this by bringing the nodes next to opening sides and snapping the to
  the side. User “snap to nearest” for this operation. Then select and delete
  the cells that are inside the opening. The mesh editing for the opening is
  illustrated in Figs 9-7 through 9-10.
Using “snap to end” edit the common cell nodes next to the end of lines to snap to the line ends (Fig. 9-10). Then, using “snap to nearest” adjust the cells that straddle across the beam or wall line to have their boundary on the line.

**FIGURE 9-4 SELECTION OF THE FOUR CELLS ADJACENT TO A COLUMN POINT**

**FIGURE 9-5 DRAGGING OF THE COMMON NODE OF THE FOUR CELLS AND THEIR SNAPPING ON THE COLUMN POINT**
FIGURE 9-6 COLUMN POINT IS COVERED BY THE COMMON NODE OF ITS ADJACENT FOUR CELLS

FIGURE 9-7 THE COMMON NODES NEXT TO A CORNER OF THE OPENING ARE PICKED AND SNAPPED TO THE CORNER
FIGURE 9-8 NODES ADJACENT TO THE OPENING ARE SELECTED, DRAGGED AND SNAPPED TO THE OPENING SIDE AND CORNER

FIGURE 9-9 EDITING OF CELLS TO ACCOMMODATE THE OPENING

(a) Opening Adjustment at Interim Stage
(b) Completion of Mesh Editing at Opening
Once the meshing is complete, the program automatically generates the shell and frame elements for the beams, columns and the walls (Fig. 9-11). The generation of these elements is achieved prior to the Finite Element Analysis of the floor system. Once the analysis is complete, the user can view the meshing of the structure in three dimensions.

An alternative to the manual meshing is automatic meshing. The automatic meshing option for the current example is shown in Fig. 9-12. The cell size selected for the automatic meshing was the same used for the manual example.
The manual meshing described herein is of great advantage, when due to the complexity of a structure, the automatic meshing fails either fully, or results in a very dense mesh.

![Meshing Using Automatic Meshing Option](image)

**FIGURE 9-12  MESHING USING THE AUTOMATIC MESHING OPTION**

### 9.2 Generation Of Triangular Cells

Quadrilateral shell elements yield a more accurate solution. However, in some instances, the geometry of a slab location is more amenable to triangular elements. For these cases triangular cells are used.

A triangular cell is created by transforming a quadrilateral cell into a triangle. This is achieved by clicking on one corner of the quadrilateral, dragging it over and snapping it onto another corner of the same quadrilateral. **Figure 9-13** illustrates the process. The quadrilateral cell ABCD shown in part (a) of the figure is transformed into the triangular cell ABD. The following are the steps.

1. Activate the snap to intersection tool and turn other snap tools off
2. Select the corner to be eliminated (point C in the figure)
3. Snap the corner selected onto another vertex of the quadrilateral (point D)
Figure 9-14(a) shows the plan of an irregular Slab Region. Selection of a 6 by 6 division when snapped at the four convex corners of this Slab Region give a mesh as shown in part (b) of the figure. Note that the cell falling entirely outside the boundary of the Slab Region is not generated by the program. This is a feature designed to minimize the effort of manual meshing by the user.

Figures 9-15 through 9-19 show graphically how the nodes A and B of the quadrilateral cell next to corner C of the slab are edited to conform with the geometry of the slab boundary. The figures also illustrate the editing of the adjacent cells to complete the meshing along the slab boundary line CD.
FIGURE 9-15 CLOSE UP OF THE TOP LEFT CORNER OF THE SLAB

FIGURE 9-16 NODE A IS DRAGGED AND SNAPPED ON LINE CD
FIGURE 9-17  NODE B IS DRAGGED AND SNAPPED AT THE INTERSECTION OF A ON CD

FIGURE 9-18  TOP LEFT QUADRILATERAL CELL IS TRANSFORMED TO A TRIANGULAR CELL MATCHING THE SLAB GEOMETRY
FIGURE 9-19  EXCESS CELLS ARE REMOVED AND THE REMAINDER
EDITED TO MATCH THE SLAB BOUNDARY

10. Glossary of Terms

10.1 Natural Nodes

Each structural component will have a number of “natural nodes” when converted
into finite elements for analysis. The natural node is located at the centroid and end-
section of the frame elements. Hence, each frame element has two natural nodes, one
at each end. For shell elements, the natural nodes are at each corner of the shell
element and mid-depth of its thickness. The natural nodes are located on the midplane
of the shell element. Figure 10.1-1 illustrates several structural components and the
position of the natural nodes.

10.2 Analysis Nodes

Unlike the traditional finite element formulations, in order to handle post-tensioning
and also to take advantage of the features of component technology, ADAPT’s
formulation is based on “analysis nodes.” Each finite element of the structure is
mapped onto a reference plane as illustrated in Fig. 10.2-1. The image of a natural
node on the reference plane is its “Analysis Nodes.” For example, I’ is the analysis
node of the natural node I. ADAPT’s calculations are performed on the analysis
nodes. However, the solution is reported and displayed with respect to the natural
nodes. In summary, the analysis node concept is an analytical feature necessitated to
move beyond the traditional application of the Finite Element Method and use the
more intuitive design approach of “component technology.” The introduction and
description of “analysis nodes” in this document is intended to provide a smooth
informational transition from the traditional FEM to Component Technology. Otherwise, the knowledge of, and use of analysis nodes for the design engineer using ADAPT software is not necessary.
10.3 OFFSET

Offset is the distance a natural node must be shifted to reach its analysis node. Refer to Fig. 10.1-3 and note that offsets are expressed in terms of translations along the global coordinate axes. For example, OX is offset along X-direction.
FIGURE 10.3-1  NATURAL AND ANALYSIS NODES

NATURAL TO GLOBAL NODE OFFSETS