User's Manual:

mesh3d_scp

Summary

This manual describes how to generate input data for a three–dimensional finite element analysis of a plate containing a surface crack using the mesh generating program **mesh3d_scp**.

Content

1. General features

This manual gives a short description of how to generate a finite element model of a surface cracked plate using the mesh generating program mesh3d_scp. It is assumed that the cracked plate, see Figure 1, possesses two planes of symmetry both in terms of geometry and in terms of loading conditions, so that only a quarter of the model needs to be modeled. The model consists of solid brick elements comprised of 8, 20 or 27 nodes. All nodes and elements are written out explicitly on the output files. Input data can be generated for the finite element programs ABAQUS, ADINA, WARP3D. Input data can also be generated on a neutral patran format.

The model is divided into three regions: *zone S*, *zone A* and *zone B,* see Figure 2. In each region node and element numbering is controlled by three indices in a systematic way, making it easy to locate specific nodes and elements in the different parts of the model. This simplifies the definition of the boundary conditions and application of external loading.

The tubular region embracing the crack front, zone S, is formed by ellipses and hyperbolas by use of conformal mapping, described to some extent in section 2 below. The tip of the continuous crack front can be modelled in two ways: (*i*) as being sharp or (*ii*) with a small finite root radius. The latter choice is preferable in cases where finite strain effects are taken into account and the former choice should be used in analyses where small strains is assumed.

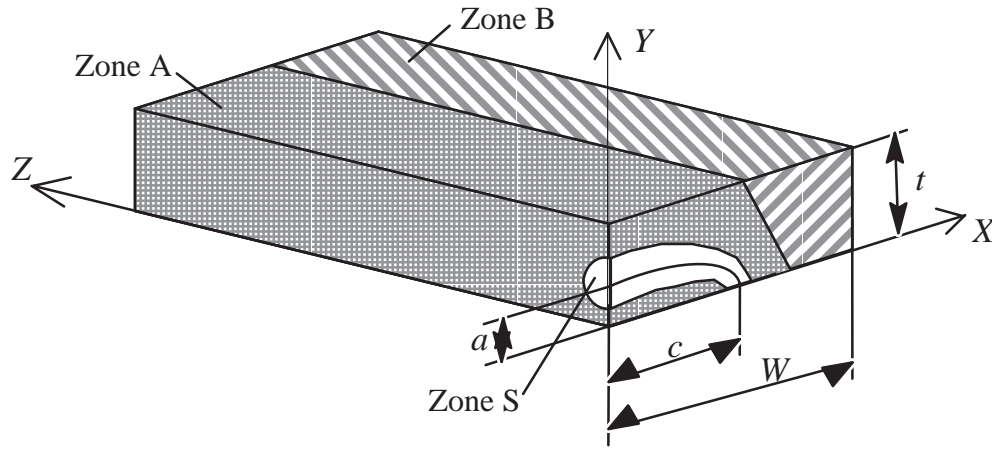Figure 1. The basic geometry of the
cracked plate.
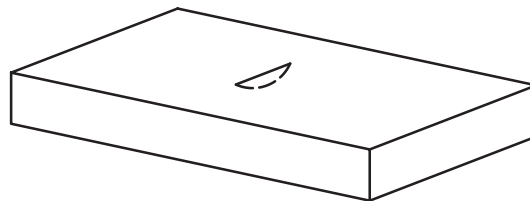
Figure 2. The model is divided into three regions, zone S, zone A and zone B. Some of the geometrical parameters needed to generate the model as well as the global coordinate system are indicated in the figure.
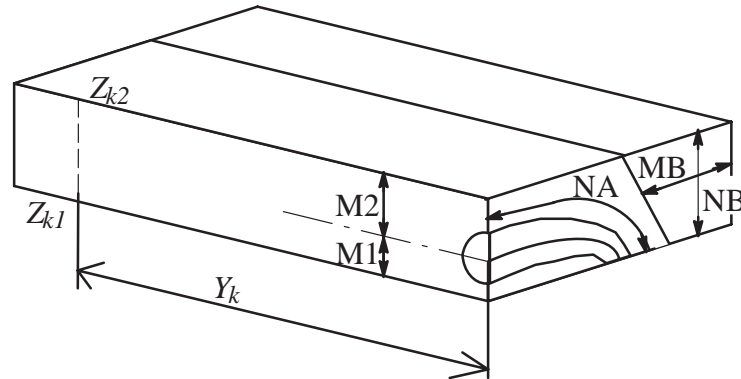

Figure 3. Showing parameters defining the number of elements in different regions of the model.

2. Input file and input data

The parameters needed to generate a model are given in the input data file "**mesh3d_scp.in**". The program starts to read the input data after the first row containing ***IN**. The input parameters are then read from the subsequent rows. In order to simplify the assignment of the data, the parameter names may be written before each parameter but the names must not contain any digits, only characters are allowed. How the input data should be given are shown below.

File: **mesh3d_scp.in**

```
*INPUT DATA
job
program
date
MR    MF      MV        SFRED
M1    M2      NA        NATYPE       ω
MB    NB      MBTYPE    MB_BIAS
LT    LRED    RTYPE
NO_OF_EL_NODES    EL_CH_ZS
R1    R2      η         μ
RN    RZERO   QUARTPNT
T     W       C         A
```
Y_0 Z_{01} Z_{02}
Y_1 Z_{11} Z_{12}
..   ..
Y_k Z_{k1} Z_{k2}
..   ..
Y_{LT} Z_{LT1} Z_{LT2}

The parameters have the following meaning:

job   the name of the job.

program   = ABAQUS, ADINA, WARP3D or PATRAN

date   date on the form yymmdd or ddmmyy or mmddyy or ... (for book–keeping only).

MR, MF, MV   defines the number of elements in the tubular region surrounding the crack front (zone S), see Figure 4.

SFRED   if > 0, mesh coarsening in *zone S* in the circumferential direction outside the ele–ment ring number equal to the parameter *SFRED*.

M1, M2, NA, MB, NB   defines the number of elements in zone A and zone B, see Figure 3.

NATYPE   type of element division/grading along the crack front, see mesh grading below.

ω	used to divide the element along the crack front, see mesh grading below.
MBTYPE	= 0 mesh grading in the *X*–direction in zone B is automatically done by an internal scheme and depends on the mesh grading in zone A. = 1 mesh grading in the *X*–direction in zone B is controlled by *MB_BIAS*.
MB_BIAS	mesh grading parameter in zone B, equal to the quotient between the element length of two subsequent elements in the *X*–direction.
LT	total number of element layers in the *Y*–direction.
LRED	number of element layers in the *Y*–direction before element reduction.
RTYPE	= 0 no element reduction in zone A and B (*LRED* has no meaning in this case). = 1 element reduction: 2 to 1 in zone A only. = 2 element reduction: 4 to 1 in zone A and 2 to 1 (*Z*–dir.) in zone B.
R1, R2, η, μ	see Figure 4.
RN	root radius at the crack tip (in the plane *X*=0, will decrease when approaching the free surface, due to conformal mapping).
RZERO	= 0 a small finite root radius (*RN*) will be employed at the crack tip. = 1 the crack tip will be modelled as sharp (*RN* has no meaning in this case).
ELAST	= 0 elastic–plastic analysis assumed—the crack tip nodes will have independent degrees of freedom. = 1 linear elastic analysis assumed—the crack tip nodes at the same crack front location are given the same node number (WARP3D) or are tied together by use of constraint equations (ABAQUS). (Note that quarter point positioning of the mid nodes is not used.)
t, W, c, a	geometry parameters, see Figure 2.
NO_OF_EL_NODES	number of nodes per element which can be 8, 20 or 27.

EL_CH_ZS	= 0: one element set is generated allowing for one element type to be used.
	> 0: two element set is generated, these are
	    (*i*) elements in the *"EL_CH_ZS"* number of element rings closest to the crack front in zone S. If 20–node elements was chosen, these elements should be fully integrated in order to avoid spurious energy modes and
	    (*ii*) elements in the remaining part of the model. In the case of 20–node elements reduced integration could be used.

SLICE	=0 if *program*=ABAQUS, WARP3D or PATRAN =0 or 1 if *program*=ADINA. If =1, elements will be divided into groups where each group makes up a layer of elements perpendicular to the crack front. This is useful during post processing of results.

Y_k, Z_{k1}, Z_{k2} defines the Y–position and the thickness of element layers according to Fig. 3.
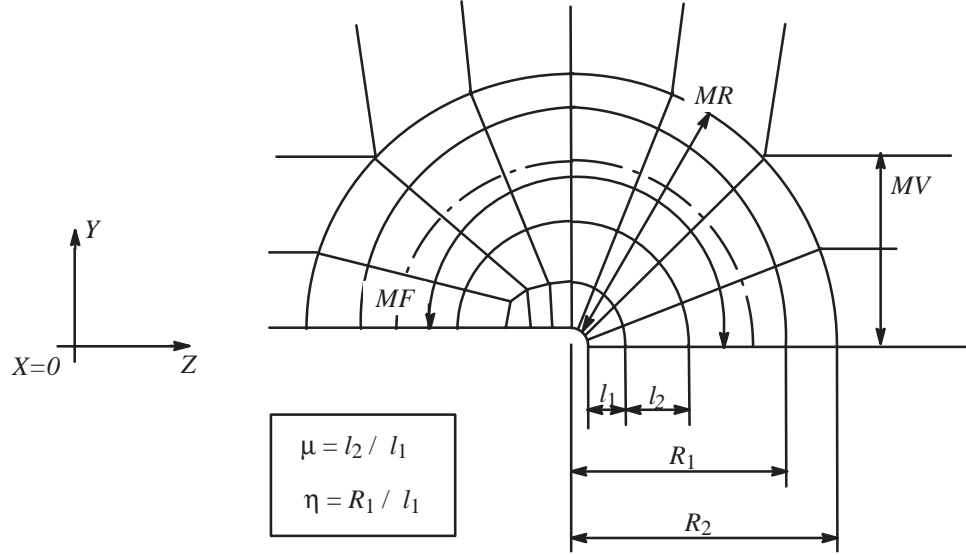

Figure 4. A schematic picture of the focussed mesh in zone S.

2.1 Mesh grading

Radial grading the focused mesh in zone S:

The radial length of the i:th element in the focused mesh region (zone S) is determined by

$$l_i = l_1 \mu^{(i-1)\beta}, \quad 1 \le i \le MR - 1, \tag{1}$$

where l_1 is the length of the element closest to the crack border $\mu = l_2 / l_1$. Both l_1 and μ should be given as input data, see above. The exponent β in (1) is determined numerically by the expression

$$R_1 = \sum_{i=1}^{i=MR-1} l_i , \tag{2}$$

where l_i is given by (1). Note that if β happens to coincide with unity, l_1, l_2, ..., l_k will corre–spond to a geometrical series.

Element grading along the crack front:

The mesh in the focused region (zone S) is, as mentioned earlier, formed by ellipses and hyper–bolas, as schematically pictured in Figure 5. The coordinates can then be evaluated according to

$$X = a\left(\rho + \frac{1}{\rho}\right)\cos\varphi ,$$

6

$$Y = \alpha\left(\rho - \frac{1}{\rho}\right)\sin\varphi,$$

where α is a scaling factor determined by the crack geometry as

$$\alpha = \frac{1}{2}\sqrt{c^2 - a^2}.$$

Thus, the sides of the elements coincide with constant values of either ρ or φ. There are three different alternatives for dividing the crack front in element segments. These alternatives are:
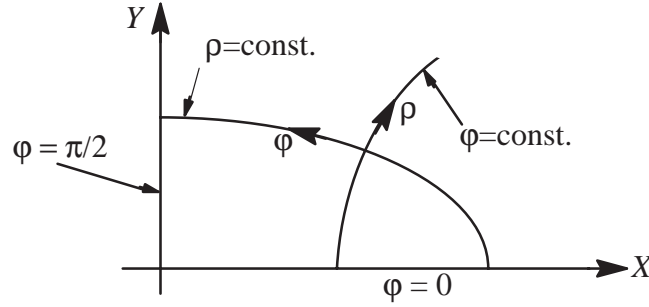

Figure 5. The mesh in zone S and zone A is generated by means of conformal mapping.

Alternative 1, NATYPE $= 1$:

$$\varphi_j = \frac{\pi}{2}\frac{(NA + 1 - j)}{NA}, \qquad 1 \le j \le (NA + 1),$$

where j is the index for the vertex nodes at the crack front, equal to 1 at the symmetry plane ($X = 0$) and equal to $2NA+1$ at the free surface.

Alternative 2, NATYPE $= 2$:

$$\varphi_j = \arcsin\psi_j, \quad 1 \le j \le (NA + 1),$$

where

$$\psi_j = \left(\frac{(NA + 1 - j)}{NA}\right)^{\omega}, \quad 1 \le j \le (NA + 1).$$

If ω which is given as input data, is equal to unity the elements along the crack front will have equal lengths in the Z–direction, *i.e.* $\Delta Z = a/NA$.

Alternative 3, NATYPE = 3:

$$\varphi_j = \frac{\pi}{2} - \left[\Delta\varphi_1 + \Delta\varphi_2 + \ldots + \Delta\varphi_{j-1}\right], \quad 1 \le j \le (NA + 1),$$

where

$$\Delta\varphi_k = \varphi_k - \varphi_{k+1}, \quad \text{and} \quad \omega = \frac{\Delta\varphi_{k+1}}{\Delta\varphi_k}, \quad 1 \le k \le NA.$$

The increments in angle $\ldots, \Delta\varphi_k, \Delta\varphi_{k+1}, \ldots$ will then constitute a geometrical series. If the mesh is desired to be more refined close to the free surface ($\varphi \to 0$), then ω should be chosen to have a value less than unity.

For all three alternatives the angle for the mid nodes is determined as the mean value of the adjacent vertex nodes.

3. Node and element sets

3.1 Node set

The node sets created during an execution of **mesh3d_scp** are listed below. However, it is only in the case of an ABAQUS analysis that all the node sets will be written on the input data file for the FE analysis.
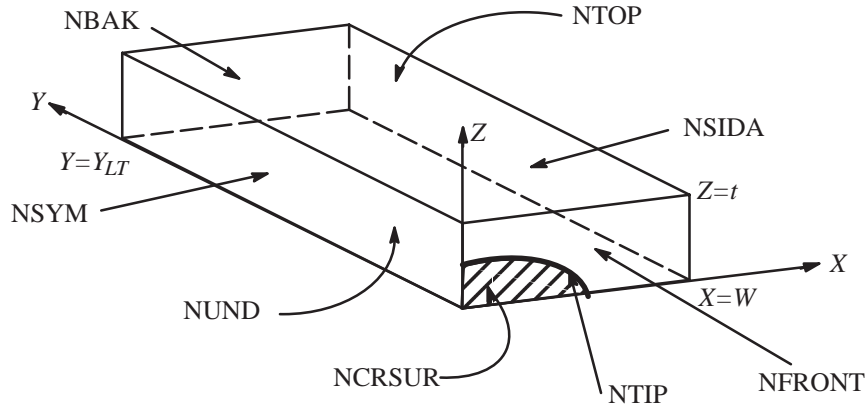
NFRONT $(Y = 0 \ \& \ \left(\frac{X}{c}\right)^2 + \left(\frac{Z}{a}\right)^2 \ge 1)$

NTIP $(Y = 0 \ \& \ \left(\frac{X}{c}\right)^2 + \left(\frac{Z}{a}\right)^2 = 1)$

NCRSUR $(Y = 0 \ \& \ \left(\frac{X}{c}\right)^2 + \left(\frac{Z}{a}\right)^2 < 1)$

NSYM $(X = 0)$

NBAK $(Y = Y_{LT})$

NUND $(Z = 0)$

NSIDA $(X = W)$

NTOP $(Z = t)$

CR01, CR02, ... , CR11, ... , CR## node sets containing the nodes in the first element ring around the crack front at one particular crack front posi– tion, used for defining J–integral contours in the case of an ABAQUS analysis (## corresponds to $2NA+1$).

3.2 Element sets

LAYERCR elements in zone S.

LAYER01 elements in layer no. 01 in the Y–direction in zone A and zone B.

LAYER02 elements in layer no. 02 in the Y–direction in zone A and zone B.

....

LAYERLT elements in layer no. LT in the Y–direction in zone A and zone B.

4. Output files of interest

ABAQUS:

(1) *job*.015 Node numbers and node coordinates.

(2) *job*.016 Element definitions (connectivity), element set 1.

(3) *job*.017 Element definitions (connectivity), element set 2, in case of two element sets.

(4) *job*.inp Input file to ABAQUS.

(5) *job*_mesh.sta Information about the model.

(6) *job*_1.dat/inf Files for generating graphical a postscript file, view $Y = 0$

(7) *job*_2.dat/inf Files for generating graphical a postscript file, view $X = 0$

(8) *job*_3.dat/inf Files for generating graphical a postscript file, view $Y = Y_{LT}$

(9) *job*_4.dat/inf Files for generating graphical a postscript file, view $Z = t$

(10) *job*_5.dat/inf Files for generating graphical a postscript file, view $Z = 0$

ADINA:

(1) *job*.15   Node numbers and node coordinates.

(2) *job*.21   Element definitions (connectivity), element set 1.

(3) *job*.22   Element definitions (connectivity), element set 2, in case of two element sets.

(4) *job*.26   Constraint equations.

(5) *job*.in   Input file to ADINA.

   *job*_mesh.sta, *job*_1.dat, ... , *job*_5.inf see above.

WARP3D:

(1) *job*.crd   Node numbers and node coordinates.

(2) *job*.elm   Element definitions (connectivity).

(3) *job*_001.const   Fixed boundary conditions

(4) *job*_001.prdsp Uniform unit displacement applied on the surface defined by the node set
                NBAK.

(5) *job*_002.prdsp Linearly varying (Z–direction) unit displacement applied on the surface
                defined by the node set NBAK.

(6) *job*.inp   Input file to WARP3D.

   *job*_mesh.sta, *job*_1.dat, ... , *job*_5.inf see above.

PATRAN:

(1) patran.out.1 definitions of nodes, elements and fixed displacement boundary conditions.

In addition all the files generated for a WARP3D analysis will also be generated.

5. Creating postscript files for viewing the model

Generating a model is often done in an iterative fashion—a set of input parameters is chosen, the program is executed and the model is graphically examined by a number of views as defined in Section 4 above. If the model doesn't "look sufficiently good" some of the parameters should be changed and the whole procedure should be repeated. The graphical examination is done in the following way:

(*i*) generate postscript files of one of the views at time by executing the program **mesh_plot**,

(*ii*) examine the generated postscript files, *job_#*.ps, using a postscript–previewer.

It is possible to *zoom* in the views. This can be done by simply changing the "viewing window" defined by X_{min}, X_{max}, Y_{max} and Y_{min} in the files *job_#*.inf (see below). The program **mesh_plot** is also described in more detail in the information file **mesh_plot.doc**.

File: *job_#*.inf

X_0 Y_0 X_{min} X_{max} Y_{min} Y_{max} X_{size} Y_{size}

X_{start_label} ΔX_{label} $NX_{tick-marks}$ Y_{start_label} ΔY_{label} $NY_{tick-marks}$ *Size_of_digits*

. . .

. . .

6. Example

An example is enclosed on the following pages , showing the input data file and the five different views created during an execution.