User’s Manual:

mesh3d_scpcell

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_scpcell**. The mesh adjacent to the crack front is generated using the concept of *computational cell element* to facilitate simulation of crack growth.

Content

1. General features

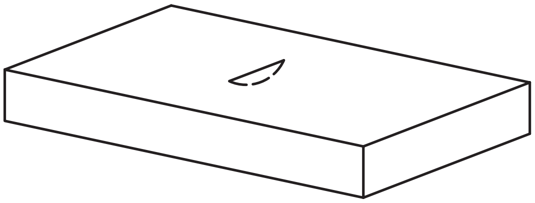
This manual gives a short description of how to generate a finite element model of a surface cracked plate using the mesh generating program **mesh3d_scpcell**. The mesh adjacent to the crack front is generated using the *computational cell element* concept, see article by Xia, Shih and Hutchinson[*] (1995), in order to facilitate simulation of crack growth. 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. An example can be seen in Figure 2.

The model consists of solid brick elements comprised of 8, 20 or 27 nodes. All nodes and elements are written out explicitly on output files. Input data can be generated for the finite element programs ABAQUS and WARP3D. Input data can also be generated on neutral patran format.

The model is divided into four regions: *zone C*, *zone S*, *zone A* and *zone B*, see Figure 3. *Zone C* contains the computational cell elements and it is in this zone crack growth is simulated. In each region node and element numbering is controlled by three indices in a systematic way in the program, 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 loads.

The tubular region, *zone S*, embracing the crack front as well as *zone C*, is formed by ellipses and hyperbolas by use of conformal mapping, described to some extent in section 2 below.

Figure 1. The basic geometry of the cracked plate.



[*] Xia, L., Shih, C.F. and Hutchinson, J.W. (1995) A Computational Approach to Ductile Crack Growth under Large Scale Yielding Conditions. *Journal of the Mechanics and Physics of Solids*, **43**, 389–413.

Figure 2. An example of a model generated with the program **mesh_scpcell**.

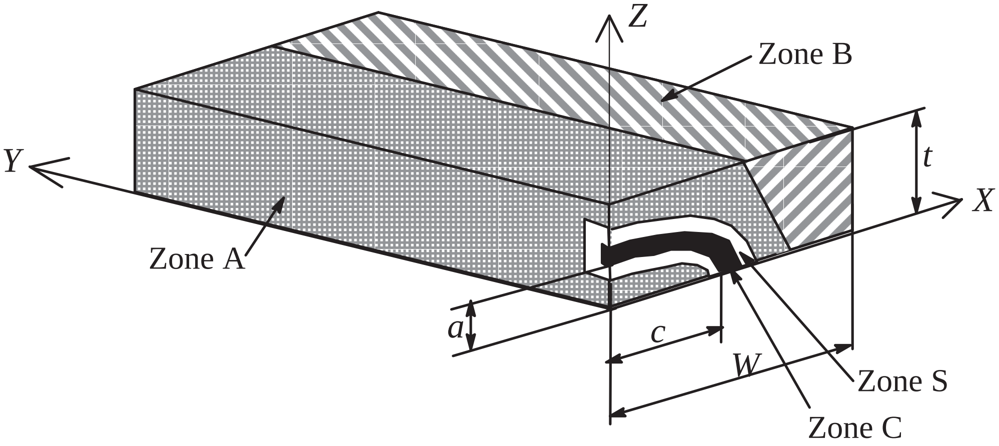


Figure 3. The model is divided into three regions, *zone C, zone S, zone A* and *zone B.* Some of the geometrical parameters needed to generate the model as well as the global coordinate system are indicated in the figure.

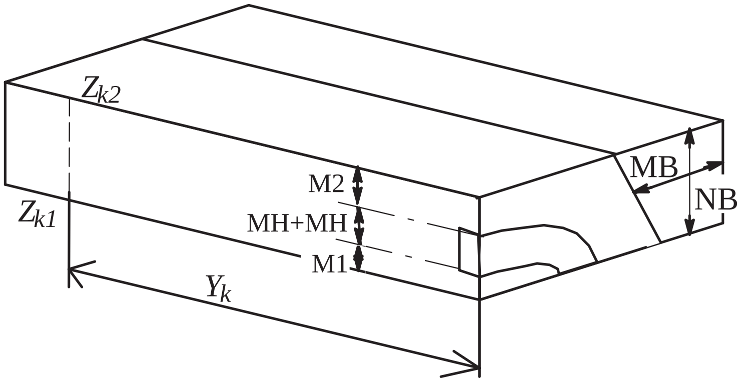


Figure 4. Showing parameters defining the number of elements in different regions of the model.

2. Input file and input data

The parameters needed to generate a model are given in the input data file “**mesh3d_scpcell.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 is shown below.

File: **mesh3d_scpcell.in**

*INPUT DATA
job
program
date
NCELL $\mathfrak{D}$ η_n η_{t1} η_{t2} *ncd1* *ncd2*
p_lc1 *p_lc2* *p_alpha*
MR *SFRED* *SFRED_TYPE* *MV* *SJRED_TYPE*
M1 *M2*
MB *NB* *MBTYPE* *MB_BIAS*
LT *LRED* *RTYPE*
NO_OF_EL_NODES
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, MV defines the number of elements in the tubular region, zone S, surrounding the region containing the computational cells which embrace the initial crack front, see Figure 5.

SFRED defines the element ring in zone S were mesh coarsening will take place and must be in the interval between 1 and *MR*, see Figure 5.

SFRED_TYPE type of mesh coarsening scheme in zone S. The available schemes are: element reduction two–to–one, *SFRED_TYPE=2* or three–to–one, *SFRED_TYPE=3.*

SJRED_TYPE mesh coarsening in the outer most element ring of zone S in the direction

tangential to the crack front. Available schemes are:
no element reduction, *SJRED_TYPE* = 1;
element reduction two–to–one, *SJRED_TYPE* = 2 (scheme used in Figure 2) or
element reduction three–to–one, *SJRED_TYPE* = 3.

NCELL number of computational cells ahead of the crack front (might be adjusted by the program).

$\mathfrak{D}$ size of the computational cell according to what is shown in Figure 5.

η_n scaled size of each of the computational cells in the crack growth direction,
if $\eta_n = 1$, the size will be equal to $\mathfrak{D}$.

η_{t1},η_{t2}, *ncd1, ncd2* used to define the mesh grading along the crack front, where $\eta_t \times \mathfrak{D}$ defines the maximum extent of a computational cell tangential to the initial crack front at the deepest point of the crack, see Figure 7 below.

p_lcx1, p_lcx2 scaled length definitions in zone S, see Figure 5.

p_alpha parameter to define the mesh in zone S. Must be given in the range of 0.1 to 0.9. An appropriate value is around 0.45.

M1, M2, MB, NB defines the number of elements in zone A and zone B, see Figure 3. Note that in the case *RTYPE* > 0, *M1+M2* must be equal to an even number.
In Figure 3 the parameter *MH* is also indicated. This parameter is automatically determined by the program to make the mesh in zone A and zone S compatible.

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 (only used if *MBTYPE*=1).

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.

t, W, c, a geometry parameters, see Figure 3.

NO_OF_EL_NODES number of nodes per element which can be 8, 20 or 27.

Y_k, Z_{k1}, Z_{k2} defines the *Y*–position and the thickness of element layers according to Fig. 4. Y_k also define the hight in zone S as is indicated in Fig. 5.

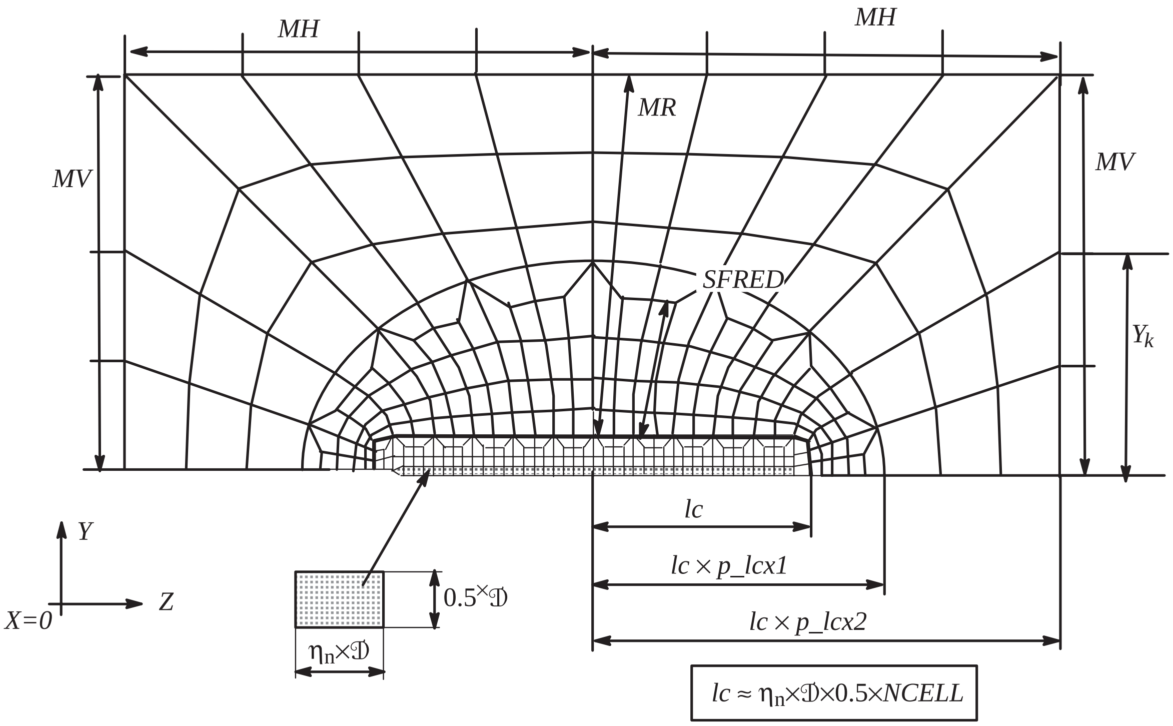


Figure 5. A schematic picture of the mesh in zone C and zone S.

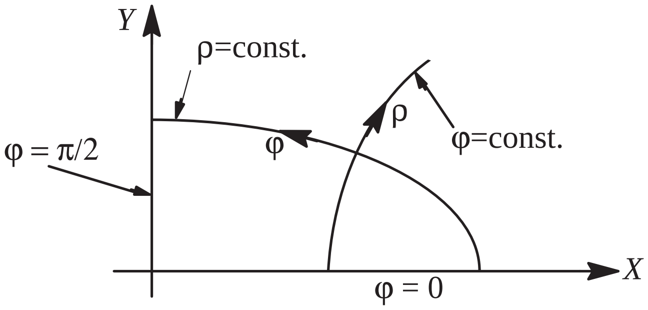


Figure 6. The mesh in zone S and zone A is generated by means of conformal mapping.

2.1 Mesh grading

Element grading along the crack front:

The mesh in zone S is, as mentioned earlier, formed by ellipses and hyperbolas, as is schematically pictured in Figure 6. The coordinates can then be evaluated according to

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

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

where α is a scaling factor determined by the size of the semi elliptical surface crack as

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

Behind the initial crack front the sides of the elements will coincide with constant values of either ρ or φ. Ahead of the the initial crack front the sides of the elements will still conicide with constant values of φ, but ρ is chosen so that the physical length scale defined by $\mathfrak{D}$ is preserved.

The mesh grading along the initial crack front is governed by a function G(φ). The function currently implemented is shown in Figure 7. In the figure the argument is replaced with the number of elements from the the free surface and lt denotes the length of a computational cell element along the initial crack front.

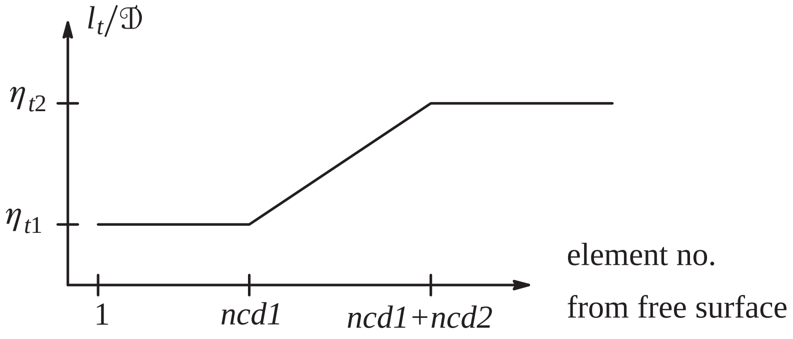
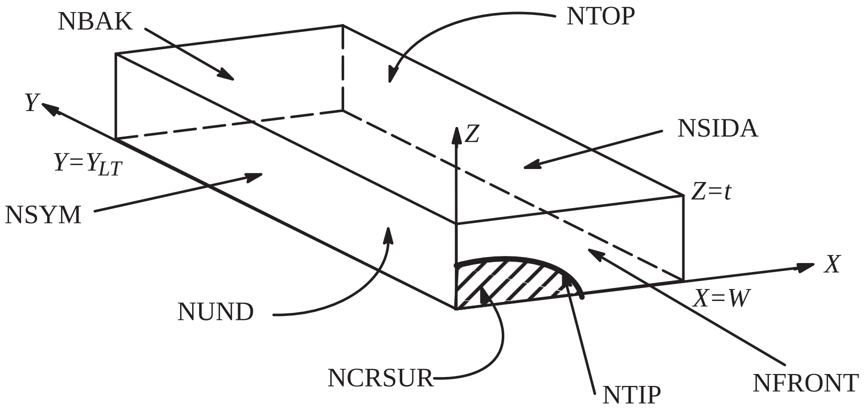


Figure 7. Mesh grading function along the initial crack front.

3. Node and element sets

3.1 Node set

The node sets created during an execution of **mesh3d_scpcell** is 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 $\left(Y = 0 \ \& \ \left(\frac{X}{c}\right)^2 + \left(\frac{Z}{a}\right)^2 \geq 1\right)$

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

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

NSYM $(X = 0)$

NBAK $(Y = Y_{LT})$

NUND $(Z = 0)$

NSIDA $(X = W)$

NTOP $(Z = t)$

CR01, CR02, ... , CR11, ... , CRnn node sets containing nodes to be used to define domains for evaluation of the *J*–integral. Only used in case of an ABAQUS analysis. nn siginfies the maximum number of points along the crack front where the *J*–integral will be evaluated.

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.

. . .

LAYER*LT* 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), Gurson elements (computational cells).

(3) *job*.017 Element definitions (connectivity), background material (normal elements).

(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 $Z = 0$

WARP3D:

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

(2) *job*_cl.elm Element definitions (connectivity), Gurson elements (computational cells).

(3) *job*_rg.elm Element definitions (connectivity), background material (normal elements).

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

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

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

(7) *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}
$X_{\text{start\_label}}$ ΔX_{label} $NX_{\text{tick–marks}}$ $Y_{\text{start\_label}}$ ΔY_{label} $NY_{\text{tick–marks}}$ *Size_of_digits*
. . .
. . .

6. Example

An example is enclosed on the following pages, showing the input data file and the five different views generated during an execution of **mesh3d_scpcell**.