CRACK MESH GENERATOR: CRACK-GEN

(November 22, 1999)

1. Output Generated By Program

The command has the form:

```
StatusFlag= 0 or 1
```

The *StatusFlag* options of 0 and 1 request:

1: during execution, create a .sta file to summarize status of mesh generation process

0: do not create the .sta file during execution

2. Target Program For Mesh Generation

The mesh generation program creates input files for various finite element programs and well known mesh pre-post processing programs. The command contains one data item and that is the name of the target program. Currently, the four programs supported include:

```
ABAQUS  WARP3D  PATRAN
```

Complete, ready to run input files are generated for the ABAQUS and WARP3D finite element programs when that option is specified. Output for PATRAN is a *neutral file* containing the element and node definitions, displacement constraints, nodal forces and element pressure loadings.

To generate both a PATRAN and ABAQUS file for the model, the mesh generation program must be executed twice with different target programs specified.

3. *ANALYSIS Command

The command has the form:

```
*ANALYSIS   type= 1, 2 or 3  eqsolver= 1 or 2
```

where the *type* number requests:

1: linear elastic analysis

2: nonlinear analysis using deformation plasticity material

3: nonlinear analysis using incremental plasticity material

The analysis *type* affects the generated, crack-front mesh as follows:

type = 1:

a single node is defined at each point on the crack front which prevents blunting. Elements incident on the crack front thus have repeated nodes in their connectivity list. The w displacement is constrained to zero for the crack front nodes.

type = 2, 3:

and mesh output for ABAQUS, WARP3D and PATRAN, multiple coincident nodes are defined at each point on the crack front. Only the front node for the θ=0 element has w=0. This is the standard model to enable crack front blunting.

The *eqsolver* number requestes use of certain equation solvers available in finite element codes:

1: direct sparse solver if available

2: conjugate gradient solver if available

4. *MATERIAL Command

All elements in the generated model are assigned the same material properties. The model and corresponding properties vary with the analysis type. For analysis type = 1 (linear elastic), only Young's modulus and Poisson's ratio are required input. For analysis type = 2 (deformation plasticity), the additional material properties include the yield stress (σ_0), the power-law hardening exponent (n) and the Ramberg-Osgood scale factor (α). For analysis type = 3 (incremental plasticity), pairs of strain-stress values are specified to define the uniaxial response.

The *MATERIAL command consists of 2 or more lines of data. The first line has the form

```
*MATERIAL no=#
```

where # sets the number of material parameters. Lines following the *MATERIAL command have one of three forms:

analysis *type* = 1, no = 2

E = Young's modulus ny = Poisson's ratio

analysis *type* = 2, no = 4 or 5

E = Young's modulus ny = Poisson's ratio S0 = yield stress n = hardening exponent alpha = Ramberg-Osgood scale factor

For WARP3D input generator, use no = 4 and omit the alpha value (not used in WARP3D)

analysis *type* = 3, no = 2 +2**k* (*k = number of points on stress-strain curve*)

E = Young's modulus ny = Poisson's ratio S1 = e1 = S2 = e2 = S3 = e3 = ... Sk = ek = ...

(Use as many lines as necessary. For WARP3D, e = is the total strain whereas for ABAQUS, e = is the plastic part of the total strain)

5. *GEOMETRY Command

This command sets the primary geometry of the cracked specimen and specifies all key dimensions except for the crack size. The mesh generator supports three types of geometries: (1) flat plates, (2) straight cylindrical segments and (3) cylindrical elbows of constant internal radius but varying included angle. The TGE parameter indicates the specimen geometry. TGE values of 1## designate various flat plate configurations, TGE values 2## designate straight cylindrical segments and TGE values 3## define cylindrical elbows.

5.1. *Flat Plate Specimens (TGE = 101)*

For flat plate specimens, the *GEOMETRY command has the form

```
*GEOMETRY TGE=101  W=    L=    t=
```

where W, L and t denote the plate width, length and thickness, respectively. TGE must have the value 101. The various crack locations (embedded, surface, corner) are specified through the TCR parameter on the *CRACK command. For each of those configurations, W, L and t have specific interpretations. These configurations are illustrated in Figs. 2–5.

5.2. *Straight Cylindircal Specimens (TGE = 2##)*

For cylindrical specimens, the *GEOMETRY command has the form

```
*GEOMETRY TGE=2##  Rc=     L=    t=
```

where Rc, L and t denote the internal radius of the cylinder, the cylinder length and the wall thickness, respectively. Figure 6 illustrates the geometry and loading options for this configuration. The specified value of TGE defines the type of crack in the cylinder, as illustrated in Fig. 7. All cracks are semi-elliptical, surface breaking cracks. The TGE values currently available include:

201: internal axial crack

202: external axial crack

203: internal circumferential crack

204: external circumferential crack

5.3. *Cylindrical Elbow Specimens (TGE = 3##)*

For cylindrical elbow specimens, the *GEOMETRY command has the form

```
*GEOMETRY TGE=3##    Rc=   t=    Rp=   phi_o=
```

where Rc defines the internal radius of the cylinder, t defines the wall thickness, Rp defines the centerline radius of the elboa and *phi_o* defines the angle turned by the elbow (radians). Figure 8 illustrates the geometry of the elbow specimen and the key dimensions. The specified value of TGE defines the type of crack in the elbow. All cracks are semi-elliptical, surface breaking cracks. Figures 9 and 10 illustrate these crack configurations. The TGE values currently available include:

301: top, internal axial crack

302: top, external axial crack

303: top, internal circumferential crack

304: top, external circumferential crack

305: bottom, internal axial crack

306: bottom, external axial crack

307: bottom, internal circumferential crack

308: bottom, external circumferential crack

6. *CRACK Command

This command sets the crack location (for flat plate models) and the crack size for all models. The command has the form:

```
*CRACK TCR = #   a =   c =  keyhole =
```

where TCR sets the crack location for flat plate models and has the value 1–4. For cylinder and elbow models, TCR should be set to 2. Refer to later sections of this guide for a detailed description of the crack configurations implied by values of TCR.

The parameters a and c define the crack depth and half–length, respectively. Refer to subsequent descriptions of each crack configuration of the geometric definition of these two values.

The *keyhole* parameter defines the configuration of elements-nodes incident on the crack front. *keyhole* =0 requests a “collapsed” front where nodes are coincident; *keyhole* =1 requests inclusion of a small "keyhole" radius at the crack front. The radius is a small fraction of the element size at the front.

7. *LOAD Command

This command defines loading applied to the generated model. Three types of loading are available: (1) imposed displacements-rotations on remote boundaries, (2) imposed forces-moments on remote boundaries and uniform internal pressure (including crack faces where appropriate), and (3) variable pressure applied only to the crack face. Positive directions of displacements, forces and moments follow the global coordinate system defined for each geometry. The command has the form:

```
*LOAD TBC = 1, 2, or 3  < loading parameters >
```

where the TBC specifies the loading type as

1: prescribed displacement on remote boundaries

2: prescribed forces/moments on remote boundaries and internal pressure

3: prescribed tractions on crack face

7.1. *Imposed Remote Displacements (TBC = 1)*

The command has the form:

```
*LOAD TBC = 1  u_x =   u_y =   u_z =   phi_x =   phi_y =   phi_z =
```

where *u_x, u_y* and *u_z* denote imposed displacements on the remote end (opposite the crack) and *phi_x, phi_y* and *phi_z* denote imposed rotations (in radians). Rotations are imposed through linearly varying axial displacements over the remote end. All displacement and rotation values listed above must be specified in the command (even those with a zero value). However, not all displacements and rotations are compatible with the various geometries and crack locations. Table 1 summarizes the current compatibilities.

7.2. *Imposed Remote Forces/Moments and Internal Pressure (TBC = 2)*

The command has the form:

```
*LOAD TBC = 2  f_x =  f_y =  f_z =  m_x =  m_y =   m_z =      int_press =
```

where *f_x, f_y* and *f_z* denote imposed forces on the remote end (opposite the crack) and *m_x, m_y* and *m_z* denote imposed moments on the remote end. The forces and moments denote *total* values acting on the end of the model, irrespective the model symmetries. If *f_x* = 1000, for example, then 500 units of force are applied over a geometry/crack configuration requiring a half-symmetric model. The forces and moments act the in global coordinate system for the configuration. The mesh generator converts the specified forces and moments into axial tractions acting over the remote end using simple bending theory. The tractions are then converted work equivalent nodal forces on the remote end.

For internal cracks in the cylinder and elbow geometries, the pressure loading acts over the crack face. Similary the load generator also puts a net axial force on the end of the cylinder and elbow equal in magnitude to the pressure $\times$ inside end area of the cylinder/ elbow. This force is added to the axial force specified above by the analyst.

All force, moment and pressure values listed above must be specified in the command (even those with a zero value). However, not all components are compatible with the various geometries and crack locations. Table 2 summarizes the current compatibilities.

The loading generator also imposes zero displacements as needed to enforce the symmetry boundary conditions for each configuration.

7.3. *Imposed Crack Face Pressures (TBC = 3)*

It is often convenient to apply a variable pressure loading on the crack face to simulate the effects of residual stresses, for example. Superposition arguments are then used to find the stress intensity factor for the complex stress state in the uncracked body using equivalent pressure loads on the crack face. The command has the form:

```
*LOAD TBC = 3  p0 =  p1 =  p2 =  p3 =   q1 =   q2 =          q3 =
```

where the p and q terms are coefficients of a parametric cubic function to describe the spatial variation of pressure over the crack face (see Fig.)

The parametric coordinate η is in the crack depth direction, and ξ is along the crack surface. The coefficient *p0* defines a constant pressure over the crack face. For example, if the crack face pressure varies only on the crack depth, set the *q1, q2, q3* values to zero, or if the pressure varies only along the crack width, set the *p1, p2, p3* values to zero. If the crack face traction is constant set *p0* = constant value and *p1, p2, p3, q1, q2, q3* values all to zero.

8. *MESH Command

All elements of the generated models have the same type. This command sets the element type and has the form:

```
*MESH etyp=#  mr=#  mf=#
```

where # defines the number of element nodes. The mesh generator executes reliably at present only for 8-node brick elements, i.e., *etyp = 8*.

The optional parameters *mr* and *mf* define the number of elements in the focused mesh around the crack front, i.e. in each plane normal to the crack front. *mr* sets the number of elements in the radial direction and has a default value of 5. *mf* sets the number of elements in the circumferential (theta) direction and has a default value of 10. Note: *only* even values may be assigned to *mf*.

Upper limits:

mr the generated mesh outside the focused crack front region is not very sensitive to changes in *mr*. The generator can fail when *mr* exceeds 15-17.

mf the generated mesh outside the focused crack front region is affected by changes in *mf*. If *mf* is increased the outer mesh is also refined. Depending on the specific geometry, the mesh generator can fail when *mf* exceeds approximately 40.

9. General Input Syntax

The mesh generator has a flexible input command system. All lines in the input file following the *MESH command are treated as comment lines and ignored.

The keywords proceeding data values, i.e., *u_x* in the imposed displacement loadings are not significant to the program. Only the number and order of data values of the line has significance.

No comment lines may be interspersed within the data lines.

10. Physical Uniits

The mesh generator has no built-in physical units. The user assumes responsibility to provide input parameters with consistent physical units.

11. Examples

Three example input files for the mesh generator are presented below. In each case, creation of the Patran neutral file is requested from the mesh generator.

11.1. *Flat Plate Crack (Figure 3)*

```
StatusFlag = 1
PATRAN
*ANALYSIS   type=2
*MATERIAL   no=2
            E=30000   ny=0.30
*GEOMETRY   TGE=101 W=5 L=10 t=0.5
*CRACK      TCR=2 a=.25  c=1.0 keyhole=1
*LOAD       TBC=2 f_x=0 f_y=0 f_z=100  m_x=0 m_y=0 m_z=0 int_press=0.0
*MESH       etyp=8 mr=5 mf=10
```

11.2. *Straight Cylinder with External Circumferential Crack*

```
StatusFlag = 1
PATRAN
*ANALYSIS   type=2   eqsolver=1
*MATERIAL   no=2
            E=200.0e9  ny=0.30
*GEOMETRY   TGE=202 Rc=1.0  L=2.0  t=0.2  (cyl)
*CRACK      TCR=2   a=0.05    c=0.20  keyhole=0
*LOAD       TBC=1 u_x=0.001 u_y=0 u_z=0 phi_x=0 phi_y=0 phi_z=0
*MESH       etyp=8 mr=5 mf=10
```

11.3. *Pipe Elbow with Internal Axial Crack*

```
StatusFlag = 1
PATRAN
*ANALYSIS   type=1  eqsolver=1
*MATERIAL   no=2
            E=30000   ny=0.30
*GEOMETRY   TGE=301 Rc=10.0 t=1.0 Rp=40.0 phio=1.0
*CRACK      TCR=1 a=0.25  c=1.0  keyhole=1
*LOAD       TBC=2  f_x=0 f_y=0 f_z=100  m_x=0 m_y=0 m_z=0 int_press=100
*MESH       etyp=8 mr=5 mf=10
```

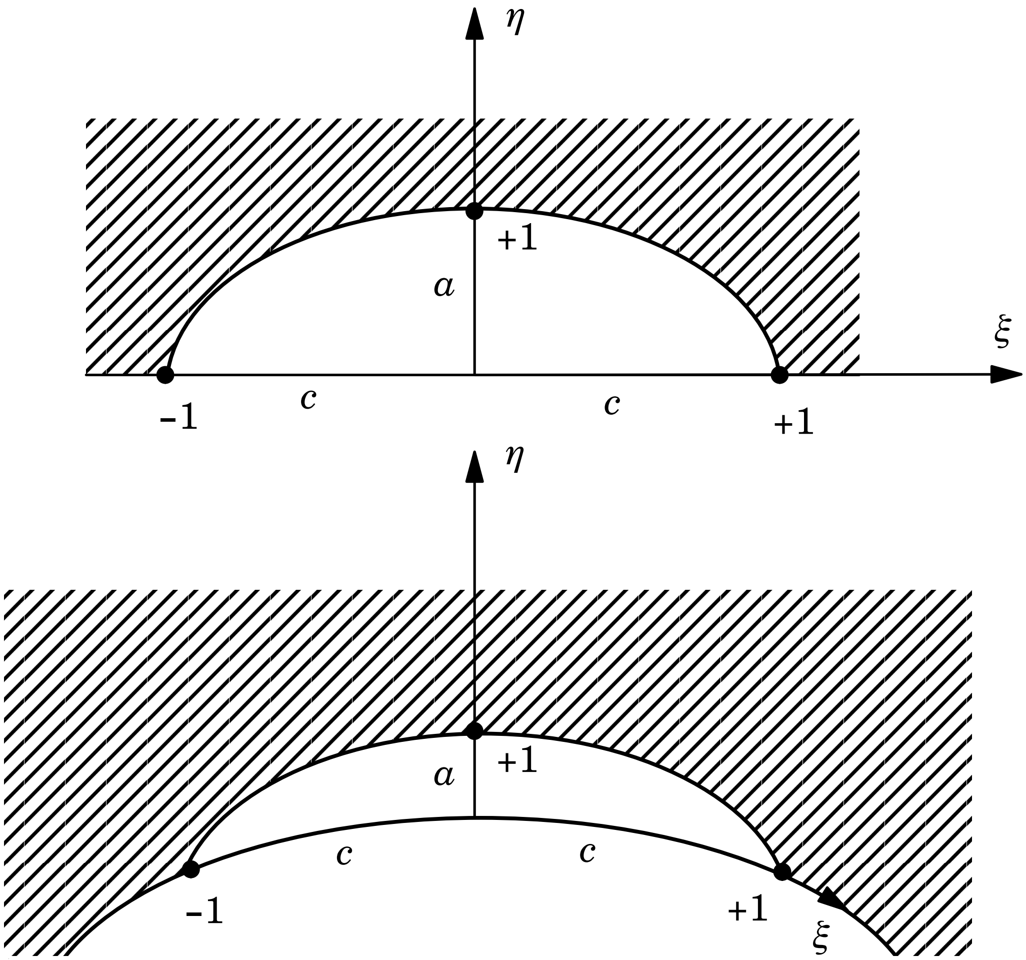
Table 1, Compatibilities Between Crack Models and Imposed Displacement Loadings

TGE	u_x	u_y	u_z	phi_x	phi_y	phi_z
101 (TCR = 1)	n.a.	n.a.	√	n.a.	n.a.	n.a.
101 (TCR = 2)	n.a.	n.a.	√	√	n.a.	n.a.
101 (TCR = 3)	n.a.	n.a.	√	n.a.	√	n.a.
101 (TCR = 4)	n.a.	n.a.	√	√	all but Abaqus	n.a.
201	√	n.a.	n.a.	n.a.	n.a.	√
202	√	n.a.	n.a.	n.a.	n.a.	√
203	√	n.a.	n.a.	n.a.	n.a.	√
204	√	n.a.	n.a.	n.a.	n.a.	√
301	n.a.	n.a.	n.a.	n.a.	n.a.	√
302	n.a.	n.a.	n.a.	n.a.	n.a.	√
303	n.a.	n.a.	n.a.	n.a.	n.a.	√
304	n.a.	n.a.	n.a.	n.a.	n.a.	√
305	n.a.	n.a.	n.a.	n.a.	n.a.	√
306	n.a.	n.a.	n.a.	n.a.	n.a.	√
307	n.a.	n.a.	n.a.	n.a.	n.a.	√
308	n.a.	n.a.	n.a.	n.a.	n.a.	√

Table 2, Compatibilities Between Crack Models and Imposed Force Loadings

TGE	f_x	f_y	f_z	m_x	m_y	m_z	int_press
101 (TCR = 1)	n.a.	n.a.	√	n.a.	n.a.	n.a.	n.a.
101 (TCR = 2)	n.a.	n.a.	√	√	n.a.	n.a.	n.a.
101 (TCR = 3)	n.a.	n.a.	√	n.a.	√	n.a.	n.a.
101 (TCR = 4)	n.a.	n.a.	√	√	√	n.a.	n.a.
201	√	n.a.	n.a.	n.a.	n.a.	√	√
202	√	n.a.	n.a.	n.a.	n.a.	√	√
203	√	n.a.	n.a.	n.a.	n.a.	√	√
204	√	n.a.	n.a.	n.a.	n.a.	√	√
301	n.a.	n.a.	n.a.	n.a.	n.a.	√	√
302	n.a.	n.a.	n.a.	n.a.	n.a.	√	√
303	n.a.	n.a.	n.a.	n.a.	n.a.	√	√
304	n.a.	n.a.	n.a.	n.a.	n.a.	√	√
305	n.a.	n.a.	n.a.	n.a.	n.a.	√	√
306	n.a.	n.a.	n.a.	n.a.	n.a.	√	√
307	n.a.	n.a.	n.a.	n.a.	n.a.	√	√
308	n.a.	n.a.	n.a.	n.a.	n.a.	√	√

Crack Face Pressure Loadings (TBC = 3)



$$p(\xi,\eta) = p_0 + p_1\eta + p_2\eta^2 + p_3\eta^3 + q_1\xi + q_2\xi^2 + q_3\xi^3$$

(where ξ, η are normalized coordinates)

Figure 1. Crack Face Pressure Loadings, TBC = 3

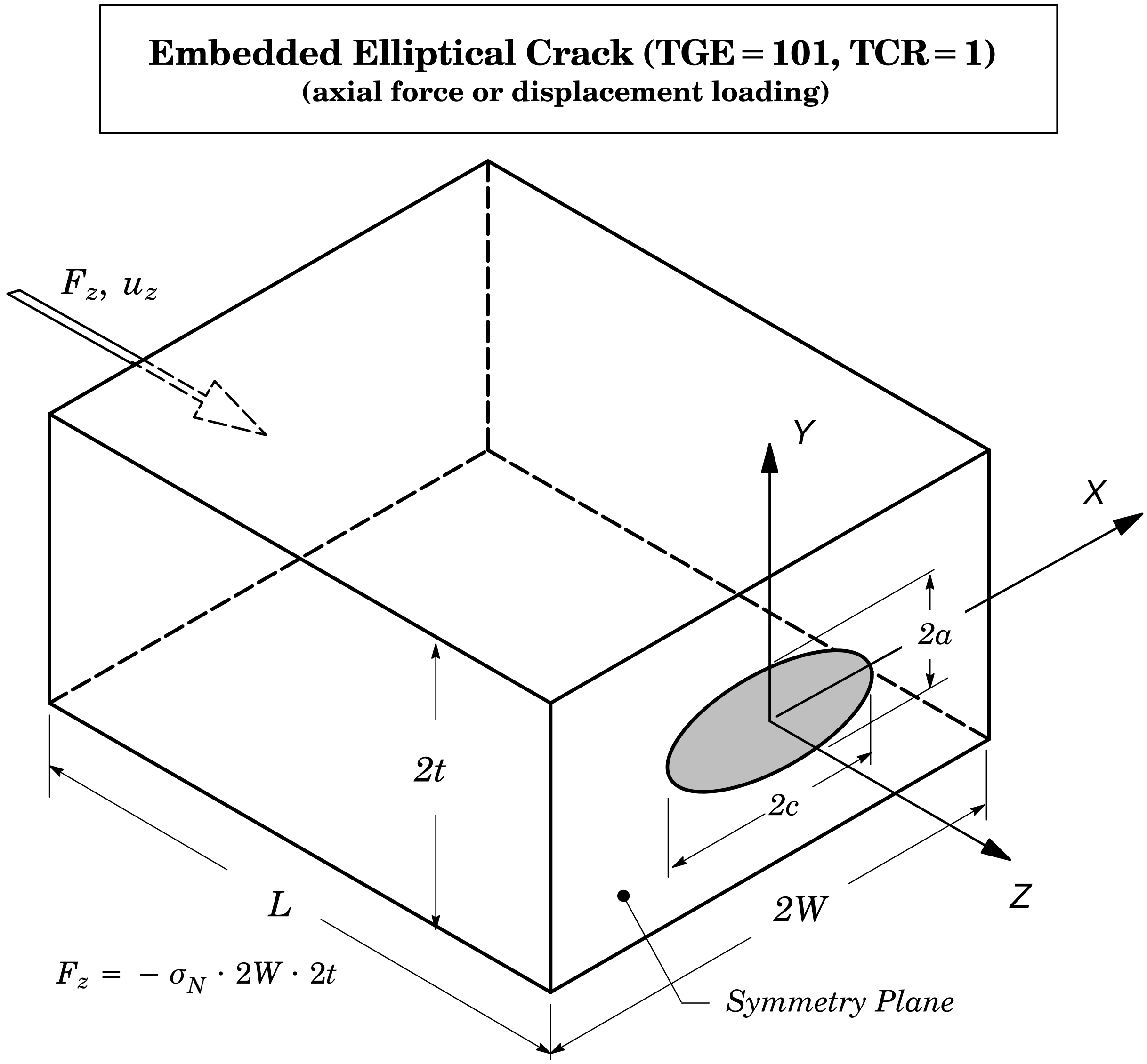


Figure 2. Embedded elliptical flaw.

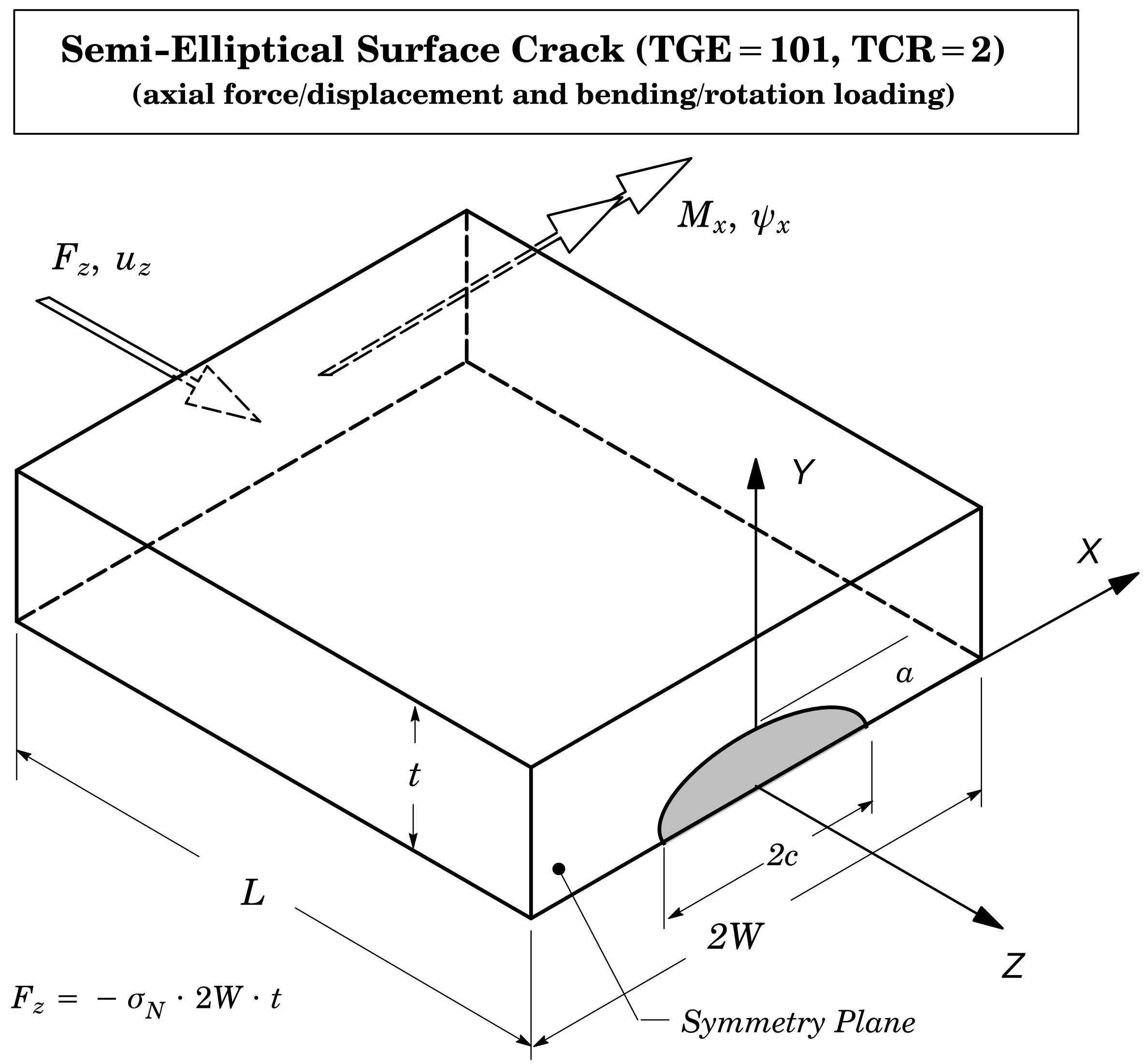


Figure 3. Semi-elliptical surface flaw.

Embedded Elliptical Crack (TGE = 101, TCR = 3)
(axial force/displacement and bending/rotation loading)

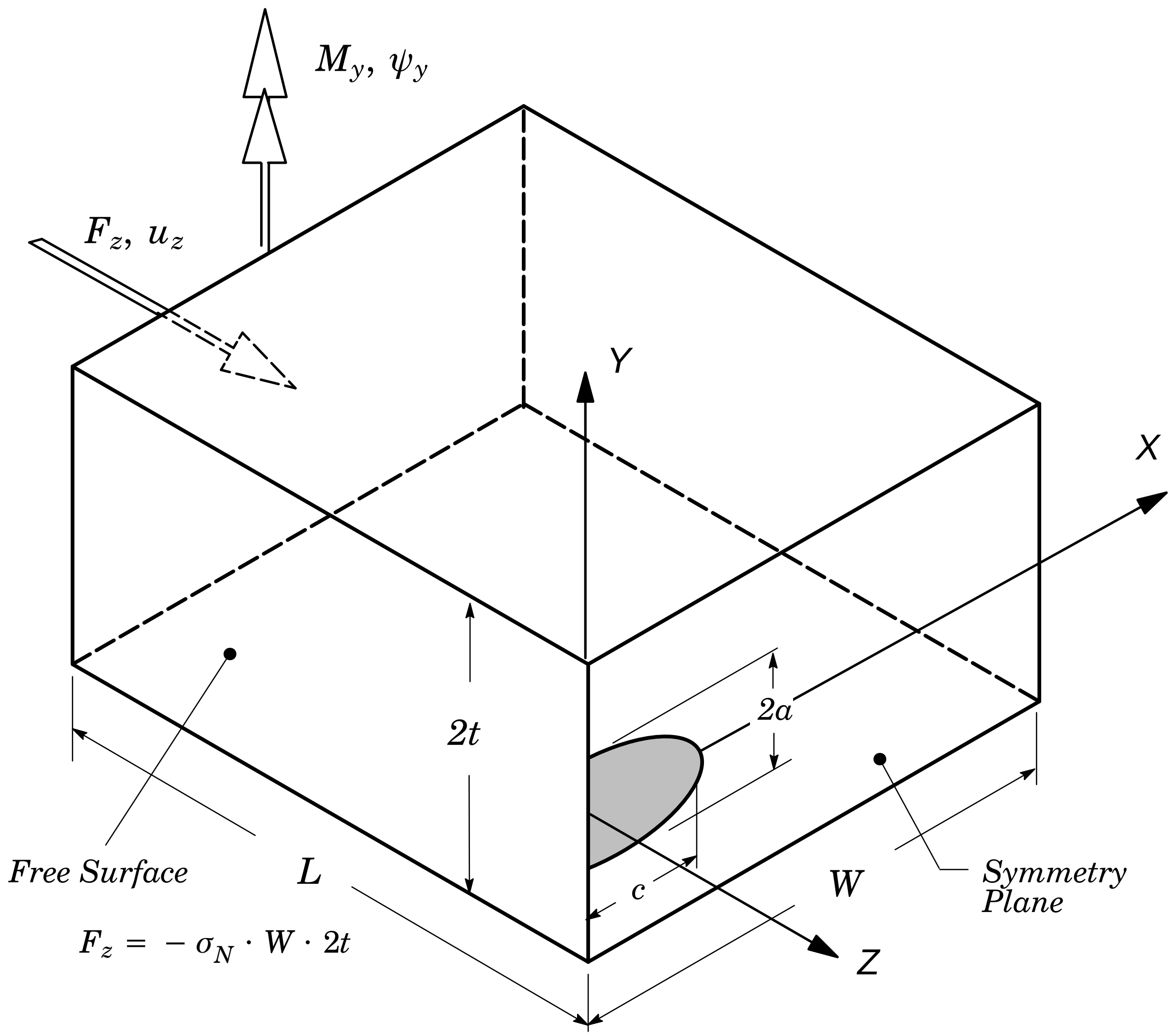


Figure 4. Semi-elliptical surface flaw.

Corner Surface Crack (TGE = 101, TCR = 4)
(axial force/displacement and bending/rotation loading)

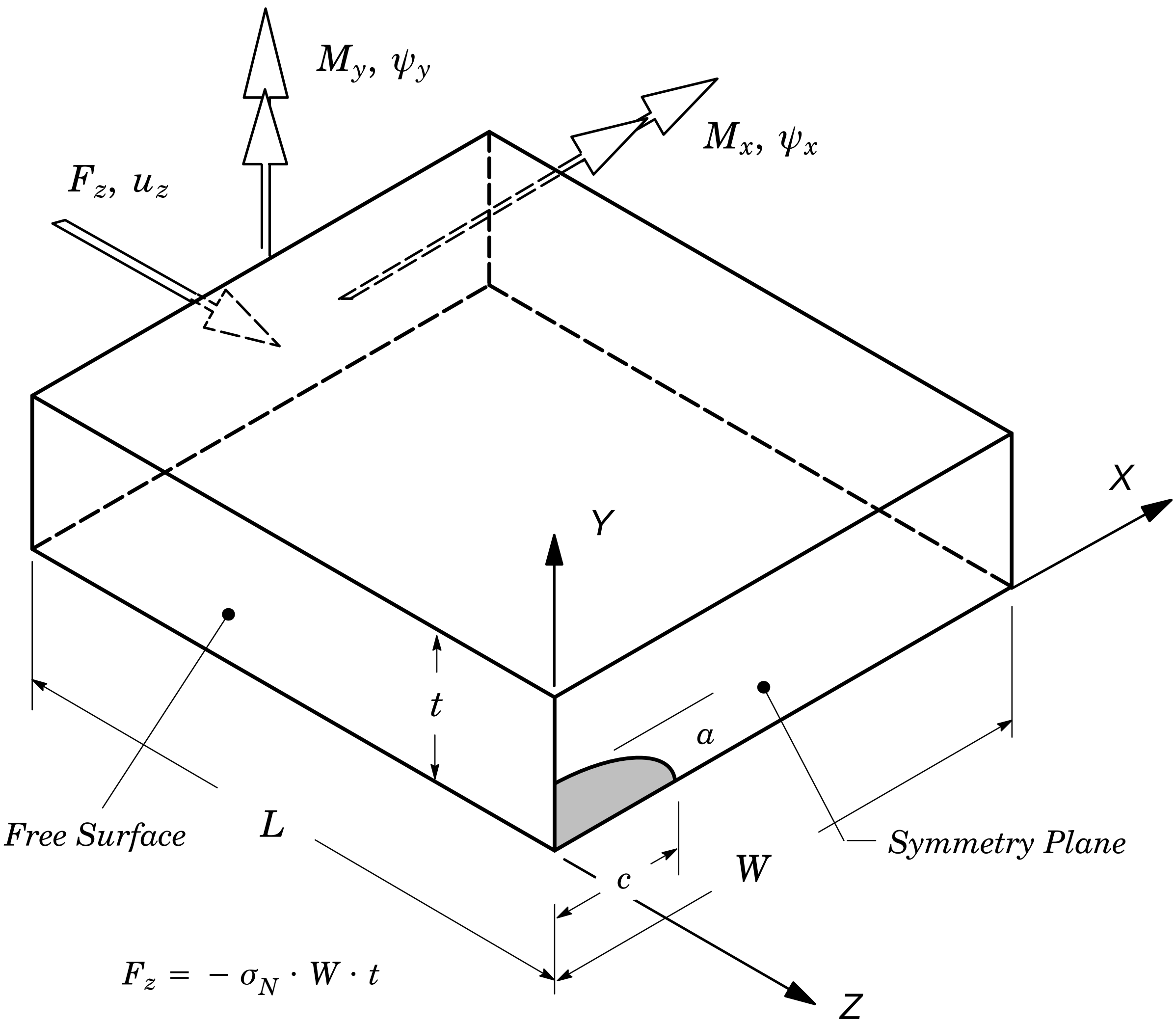


Figure 5. Semi-elliptical corner surface flaw.

Straight Cylindrical Specimen (TGE = 201-204, TCR = 2)
(geometry and loading options)

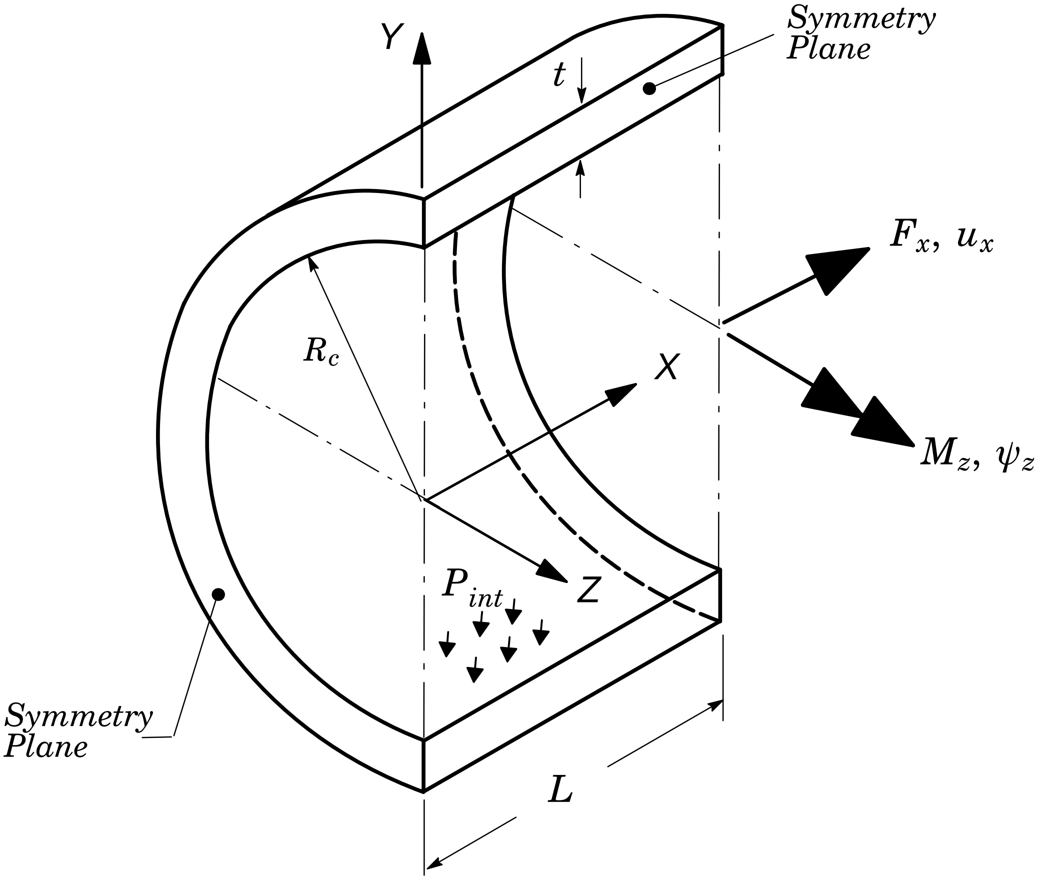


Figure 6. Straight cylindrical specimen.

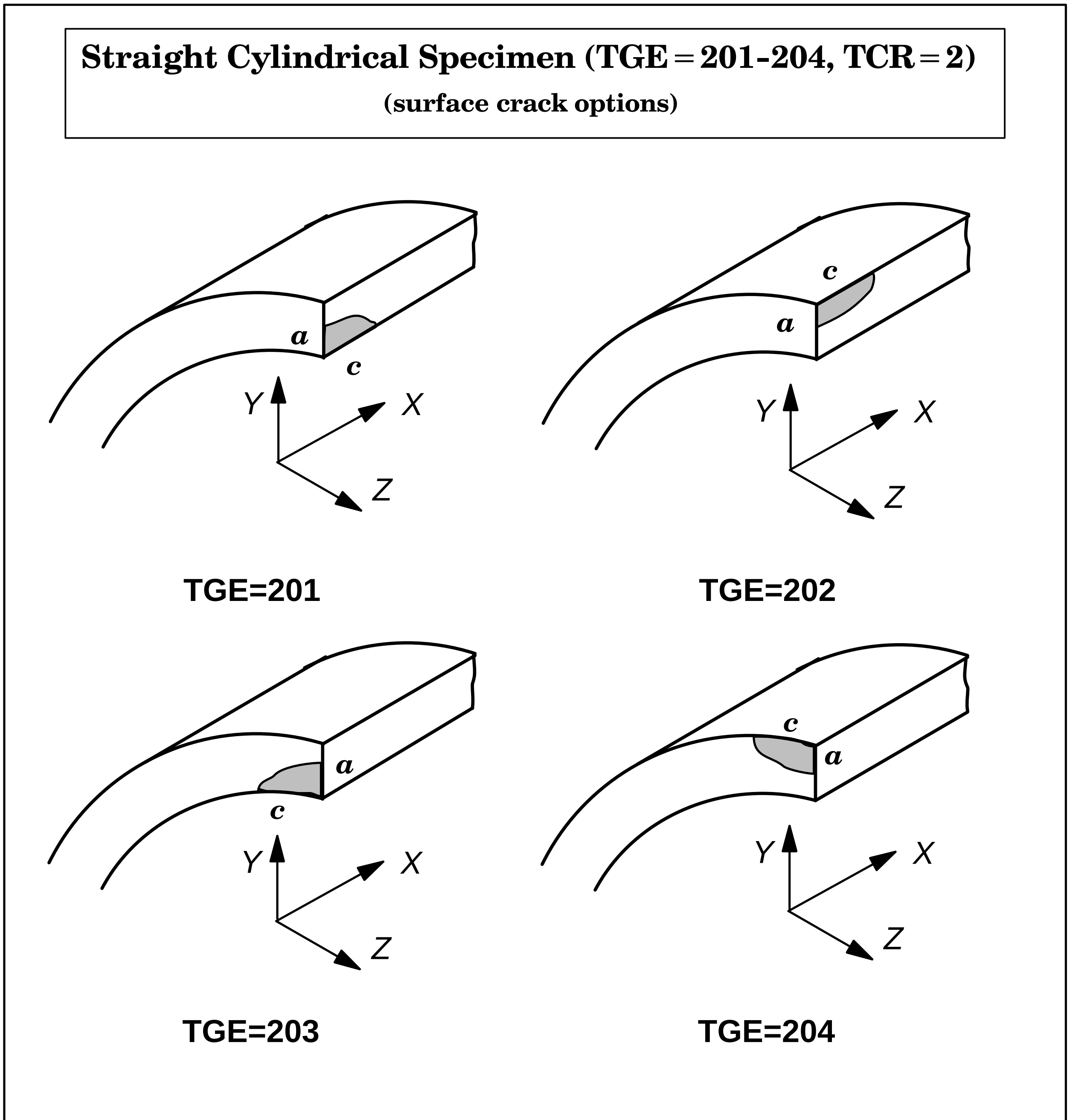


Figure 7. Straight cylindrical specimen (crack options).

Pipe Elbow Specimen (TGE = 301-308, TCR = 2)

(geometry and loading options)

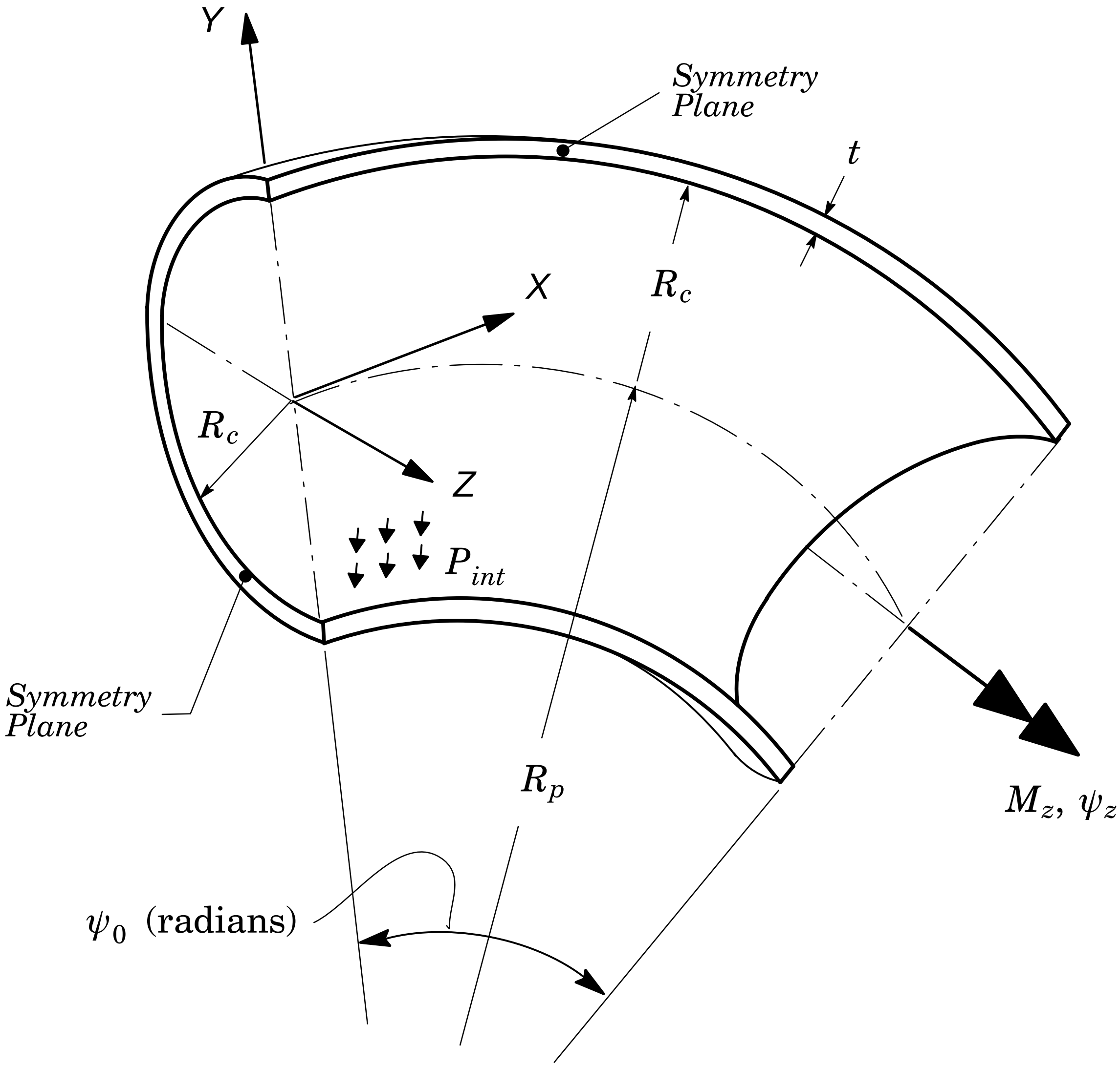


Figure 8. Pipe elbow geometry and loading.

Pipe Elbow Specimen (TGE = 301-304, TCR = 2)

(4 of the 8 surface crack options)

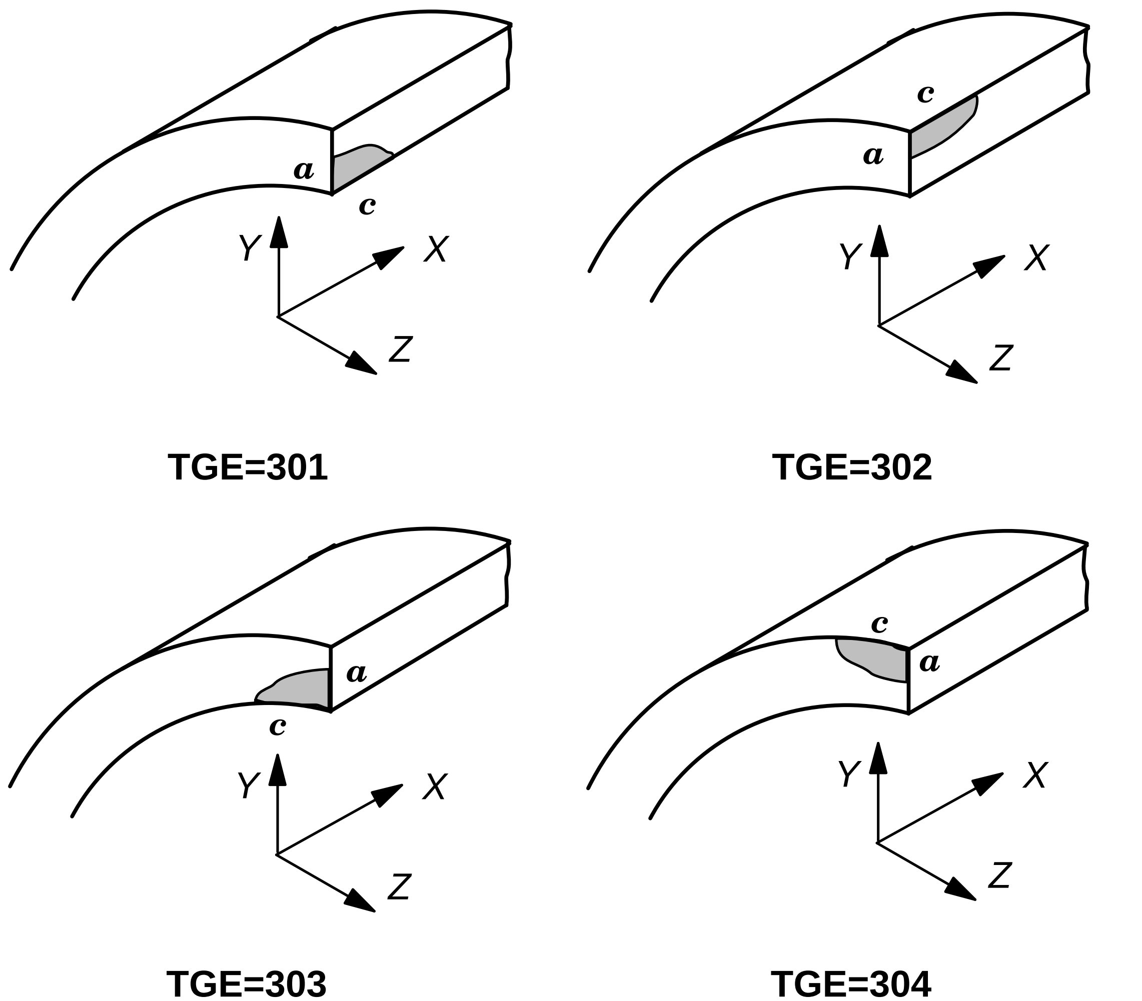


Figure 9. Pipe elbow specimen (crack options).

Pipe Elbow Specimen (TGE = 305-308, TCR = 2)

(4 of the 8 surface crack options)

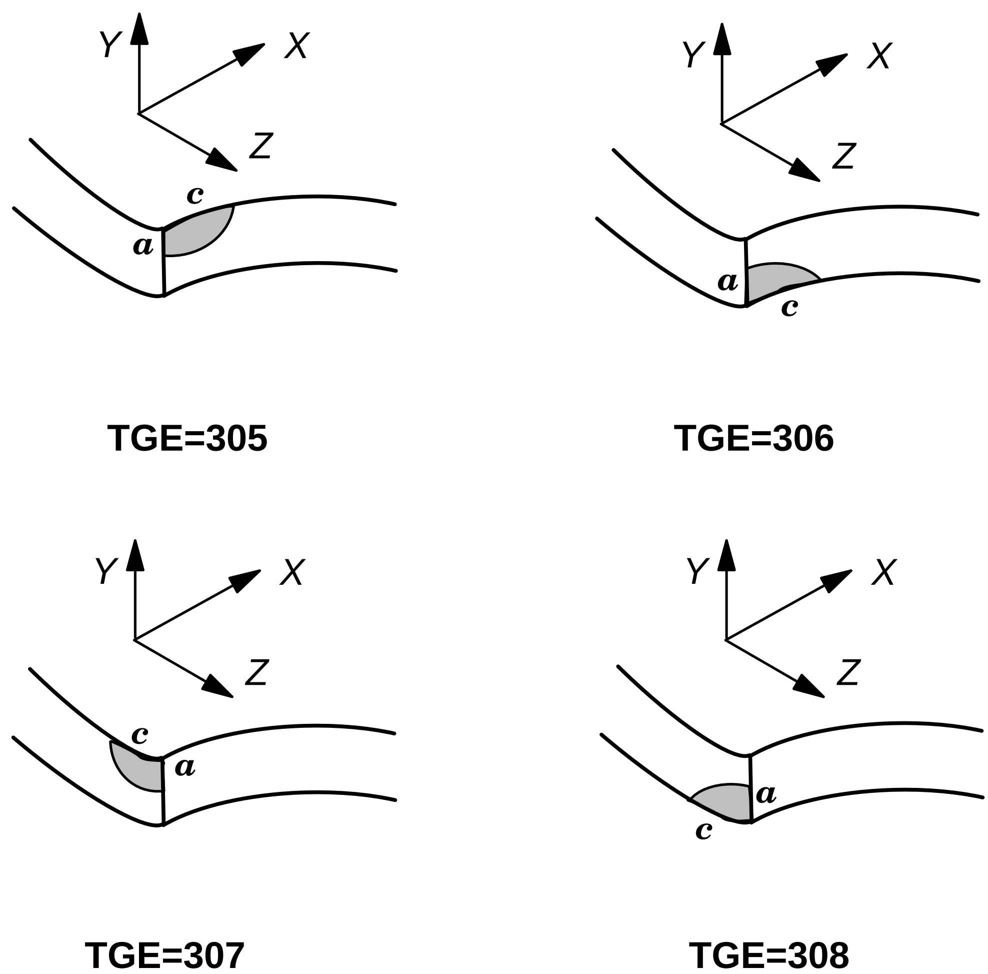


Figure 10. Pipe elbow specimen (crack options).

Appendix

Current Limitations on Model Geometry & Mesh Density
6-24-99

This program is the product of ongoing research efforts to support realistic fracture mechanics analyses of cracked components in reliable efficient ways. The mesh generator is not a commerical production code with exhaustive error checking. This appendix describes the known limitations of the current version of the mesh generator beyond which the code fails to execute properly. We have exercised the code over a variety of geometrical configurations to generate the tables below of limiting values. The study was conducted with the following parameters:

- the L/R_c ratio is held constant at 2 for the cylinders
- the R_p/R_c ratio is held constant at 4 for the elbows (additional R_c/t ratios given for reduced values of R_p/R_c)
- R_c/t ratio is increased in increments of 10
- Ratios given are the largest values that successfully produced meshes

Geometry	Largest Rc/t ratio achieved	Reason for failure at larger ratios
201 (cyl)	50:1	increased array size required
202 (cyl)	50:1	never stopped running
203 (cyl)	40:1	never stopped running/bus error
204 (cyl)	40:1	bus error/never stopped running

Geometry	Rc/t ratio	Rp/Rc	Reason for failure
301 (elbow)	30:1	4:1	traceback error
	40:1	1.75:1	(all)
302 (elbow)	30:1	4:1	traceback error
	40:1	3:1	
303 (elbow)	30:1	4:1	traceback error
	40:1	1.75:1	
304 (elbow)	20:1	4:1	traceback error
	30:1	3:1	
	40:1	1.75:1	
305 (elbow)	30:1	4:1	traceback error
	40:1	3:1	
306 (elbow)	30:1	4:1	traceback error
	40:1	3:1	
307 (elbow)	30:1	4:1	traceback error
	40:1	3:1	
308 (elbow)	30:1	4:1	traceback error
	40:1	3:1	

Analysis of 'mf' variable (refines mesh in crack theta direction)

Notes:

- the *mf* default value is 10
- when *mf* is increased by 1, 128 elements are added to the mesh
- " % inc. in elements " gives the percent increase in the total number of elements when *mf* is increased from the default value to the maximum *mf* reached

Geometry	Rc/t	mf reached	% inc. in elements	Reason for failure
201 (cyl)	50:1	11	0.25 %	traceback error
	40:1	43	11.8 %	traceback error
202 (cyl)	50:1	11	0.25 %	traceback error
	40:1	43	11.5 %	traceback error
203 (cyl)	40:1	51	11.4 %	traceback
204 (cyl)	40:1	51	11.1 %	traceback (underflow)

Geometry	Rc/t	Rp/Rc	mf reached	% inc. in elements	Reason for failure
301 (elbow)	30:1	4:1	51	11.8 %	traceback error
	40:1	1.75:1	51	11.7 %	(all)
302 (elbow)	30:1	4:1	51	11.4 %	traceback error
	40:1	3:1	10	0.0 %	
303 (elbow)	30:1	4:1	51	8.69 %	traceback error
	40:1	1.75:1	19	1.93 %	
304 (elbow)	20:1	4:1	51	16.0 %	traceback error
	30:1	3:1	51	10.1 %	
	40:1	1.75:1	51	8.53 %	
305 (elbow)	30:1	4:1	51	17.3 %	traceback error
	40:1	3:1	51	14.7 %	
306 (elbow)	30:1	4:1	51	16.8 %	traceback error
	40:1	3:1	51	14.5 %	
307 (elbow)	30:1	4:1	51	13.3 %	traceback error
	40:1	3:1	51	11.4 %	
308 (elbow)	30:1	4:1	51	13.0 %	traceback error
	40:1	3:1	51	11.3 %	

Analysis of variable 'mr' and keyhole

Notes:

- use of the keyhole (*keyhole* = 1) does not change the number of elements in the mesh (*mr* = 5, default)
- values higher of *mr* increase the number of elements in the mesh
- the default value (10) is used for the *mf* variable here

Geometry	Rc/t	Rp/Rc(for elbows)	mr reached	% inc. in elements	Reason for failure
201 (cyl)	50:1	N/A	18	21.1 %	increased array size needed
203 (cyl)	40:1	N/A	18	44.4 %	bus error/traceback
301 (elbow)	30:1	4:1	19	37.1 %	no change above 19
305 (elbow)	30:1	4:1	19	42.3 %	no change above 19