

## Element 75

## Bilinear Thick-shell Element

This is a four-node, thick-shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, displacements and the rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The transverse shear strains are calculated at the middle of the edges and interpolated to the integration points. In this way, a very efficient and simple element is obtained which exhibits correct behavior in the limiting case of thin shells. The element can be used in curved shell analysis as well as in the analysis of complicated plate structures. For the latter case, the element is easy to use since connections between intersecting plates can be modeled without tying.

Due to its simple formulation when compared to the standard higher order shell elements, it is less expensive and, therefore, very attractive in nonlinear analysis. The element is not very sensitive to distortion, particularly if the corner nodes lie in the same plane. All constitutive relations can be used with this element.

### Geometric Basis

The element is defined geometrically by the  $(x,y,z)$  coordinates of the four corner nodes. Due to the bilinear interpolation, the surface forms a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the **GEOMETRY** option.

The stress output is given in local orthogonal surface directions ( $V_1$ ,  $V_2$ , and  $V_3$ ) which, for the centroid, are defined in the following way (see [Figure 3-125](#)).

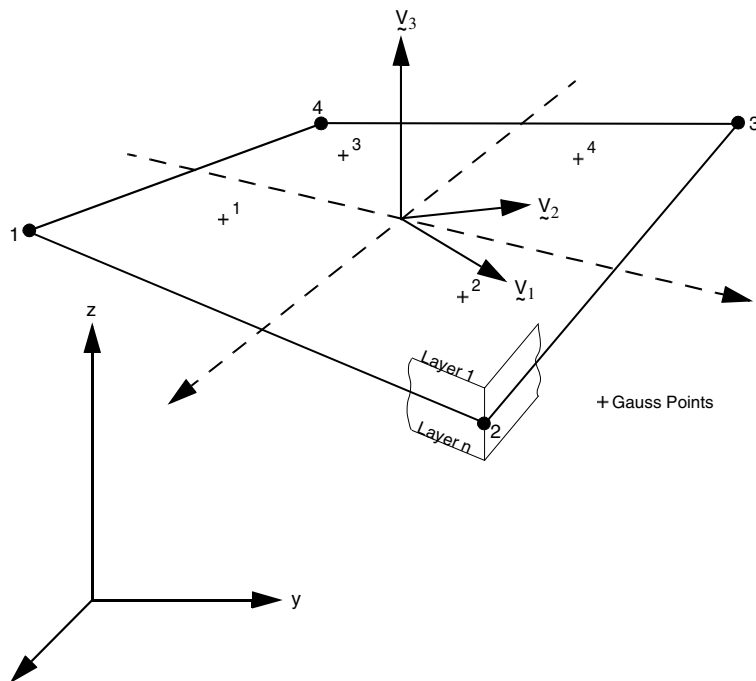


Figure 3-125 Form of Element 75

At the centroid, the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial \mathbf{x}}{\partial \xi} / \left| \frac{\partial \mathbf{x}}{\partial \xi} \right|, \quad t_2 = \frac{\partial \mathbf{x}}{\partial \eta} / \left| \frac{\partial \mathbf{x}}{\partial \eta} \right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s / \sqrt{2}|s| \quad \bar{d} = d / \sqrt{2}|d|$$

The local orthogonal directions are then obtained as:

$$\underline{V}_1 = \bar{s} + \bar{d}, \quad \underline{V}_2 = \bar{s} - \bar{d}, \quad \text{and} \quad \underline{V}_3 = \underline{V}_1 \times \underline{V}_2$$

In this way, the vectors  $\frac{\partial \mathbf{x}}{\partial \xi}$ ,  $\frac{\partial \mathbf{x}}{\partial \eta}$  and  $\underline{V}_1$ ,  $\underline{V}_2$  have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

## Displacements

The six nodal displacement variables are as follows:

$u, v, w$  Displacement components defined in global Cartesian  $x, y, z$  coordinate system.

$\phi_x, \phi_y, \phi_z$  Rotation components about global  $x$ -,  $y$ -, and  $z$ -axis, respectively.

## Quick Reference

### Type 75

Bilinear, four-node shell element including transverse shear effects.

### Connectivity

Four nodes per element. The element can be collapsed to a triangle.

### Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2 = EGEOM3 = EGEOM4 = 0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the **NODAL THICKNESS** model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -2.0. In this case, an extra line containing the offset information is provided (see the [GEOMETRY](#) option in *Marc Volume C: Program Input*). The offset magnitudes along the element normal for the four corner nodes are provided in the first, second, third, and fourth data fields of the extra line. A uniform offset for all nodes can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

### Coordinates

Three coordinates per node in the global x-, y-, and z-directions.

### Degrees of Freedom

Six degrees of freedom per node:

- 1 =  $u$  = global (Cartesian) x-displacement
- 2 =  $v$  = global (Cartesian) y-displacement
- 3 =  $w$  = global (Cartesian) z-displacement
- 4 =  $\phi_x$  = rotation about global x-axis
- 5 =  $\phi_y$  = rotation about global y-axis
- 6 =  $\phi_z$  = rotation about global z-axis

### Distributed Loads

A table of distributed loads is listed below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in $-V_3$ -direction.
3	Nonuniform gravity load per surface area in -z-direction, magnitude given in the <a href="#">FORCEM</a> user subroutine.
4	Nonuniform pressure with positive magnitude in $-V_3$ -direction, magnitude given in the <a href="#">FORCEM</a> user subroutine.
5	Nonuniform load per surface area in arbitrary direction, magnitude given in the <a href="#">FORCEM</a> user subroutine.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.
12	Nonuniform edge load magnitude given in the <a href="#">FORCEM</a> user subroutine in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction given in the <a href="#">FORCEM</a> user subroutine on 1-2 edge.
14	Uniform load on the 1-2 edge of the shell, tangent to, and in the direction of the 1-2 edge
15	Uniform load on the 1-2 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.

Load Type	Description
22	Nonuniform edge load magnitude given in the <a href="#">FORCEM</a> user subroutine in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction given in the <a href="#">FORCEM</a> user subroutine on 2-3 edge.
24	Uniform load on the 2-3 edge of the shell, tangent to and in the direction of the 2-3 edge
25	Uniform load on the 2-3 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load magnitude given in the <a href="#">FORCEM</a> user subroutine in the plane of the surface on the 3-4 edge.
33	Nonuniform edge load magnitude and direction given in the <a href="#">FORCEM</a> user subroutine on 3-4 edge.
34	Uniform load on the 3-4 edge of the shell, tangent to and in the direction of the 3-4 edge
35	Uniform load on the 3-4 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load magnitude given in the <a href="#">FORCEM</a> user subroutine in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction given in the <a href="#">FORCEM</a> user subroutine on 4-1 edge.
44	Uniform load on the 4-1 edge of the shell, tangent to and in the direction of the 4-1 edge
45	Uniform load on the 4-1 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in the <a href="#">ROTATION A</a> option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in the <a href="#">ROTATION A</a> option.

All edge loads require the input as force per unit length.

For other types of distributed loads that are normally applicable for all types of elements, please refer to [Distributed Loads](#) in Chapter 1 of this manual.

### Point Loads

Point loads and moments can also be applied at the nodes.

### Output Of Strains

Generalized strain components are:

Middle surface stretches:  $\epsilon_{11}$   $\epsilon_{22}$   $\epsilon_{12}$

Middle surface curvatures:  $\kappa_{11}$   $\kappa_{22}$   $\kappa_{12}$

Transverse shear strains:  $\gamma_{23}$   $\gamma_{31}$

in local ( $V_1$ ,  $V_2$ ,  $V_3$ ) system.

### Output Of Stresses

$\sigma_{11}$ ,  $\sigma_{22}$ ,  $\sigma_{12}$ ,  $\sigma_{23}$ ,  $\sigma_{31}$  in local ( $V_1$ ,  $V_2$ ,  $V_3$ ) system given at equally spaced layers though thickness. First layer is on positive  $V_3$  direction surface.

### Transformation

Displacement and rotation at corner nodes can be transformed to local direction.

### Tying

Use the [UFORMSN](#) user subroutine

### Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, you have to select your load steps such that the rotation remains small within a load step.

### Section Stress - Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the [SHELL SECT](#) parameter to specify the number of integration points. This number must be odd. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.g., thermal plasticity). The default is 11 points.

### Beam Stiffeners

The element is fully compatible with open- and closed-section beam element types [78](#) and [79](#).

### Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type [85](#). See Element 85 for a description of the conventions used for entering the flux and film data for this element.

### Design Variables

The thickness can be considered as a design variable.