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

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

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:

$$\underset{\sim}{V}_1 = \bar{s} + \bar{d}, \; \underset{\sim}{V}_2 = \bar{s} - \bar{d}, \text{ and } \underset{\sim}{V}_3 = \underset{\sim}{V}_1 \times \underset{\sim}{V}_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}, \frac{\partial x}{\partial \eta}$ and $\underset{\sim}{V}_1, \underset{\sim}{V}_2$ 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.
ϕ_x, ϕ_y, ϕ_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 = ϕ_x = rotation about global x-axis
5 = ϕ_y = rotation about global y-axis
6 = ϕ_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 FORCEM user subroutine.
4	Nonuniform pressure with positive magnitude in -V_3-direction, magnitude given in the FORCEM user subroutine.
5	Nonuniform load per surface area in arbitrary direction, magnitude given in the FORCEM 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 FORCEM user subroutine in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction given in the FORCEM 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 FORCEM user subroutine in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction given in the FORCEM 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 FORCEM user subroutine in the plane of the surface on the 3-4 edge.
33	Nonuniform edge load magnitude and direction given in the FORCEM 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 FORCEM user subroutine in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction given in the FORCEM 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 ROTATION 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 ROTATION 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:	ε_{11} ε_{22} ε_{12}
Middle surface curvatures:	κ_{11} κ_{22} κ_{12}
Transverse shear strains:	γ_{23} γ_{31}

in local $(\underline{V}_1, \underline{V}_2, \underline{V}_3)$ system.

Output Of Stresses

σ_{11}, σ_{22}, σ_{12}, σ_{23}, σ_{31} in local $(\underline{V}_1, \underline{V}_2, \underline{V}_3)$ 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.q., 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.