- PARAM,K6ROT isn't necessary.
- The patch-to-patch connection is sufficiently accurate if the ratio of the cross sectional area to the surface patch area is between 10% and 100%, see the figure Range of Cross-sectional Area versus Element Size for the Patch-to-Patch Connection.
- With the GRIDID or ELEMID option, the cross section of the connector may cover up to eight grid points if a quadrilateral surface patch with mid side nodes is defined, see the figure Range of Cross-sectional Area versus Element Size for the Patch-to-Patch Connection.

6.5 Gap and Line Contact Elements

In NX Nastran, you can define gap and friction elements with a CGAP entry. The element coordinate system and nomenclature are shown in Figure 6-13. CID is required, if it is used to define the element coordinate system. Otherwise, the X -axis of the element coordinate system, x_{elem} , is defined by a line connecting GA and GB of the gap element. The orientation of the gap element is determined by vector $\vec{v}$ similar to the definition of the beam element, which is in the direction from grid points GA to GO or defined by (X1, X2, X3).

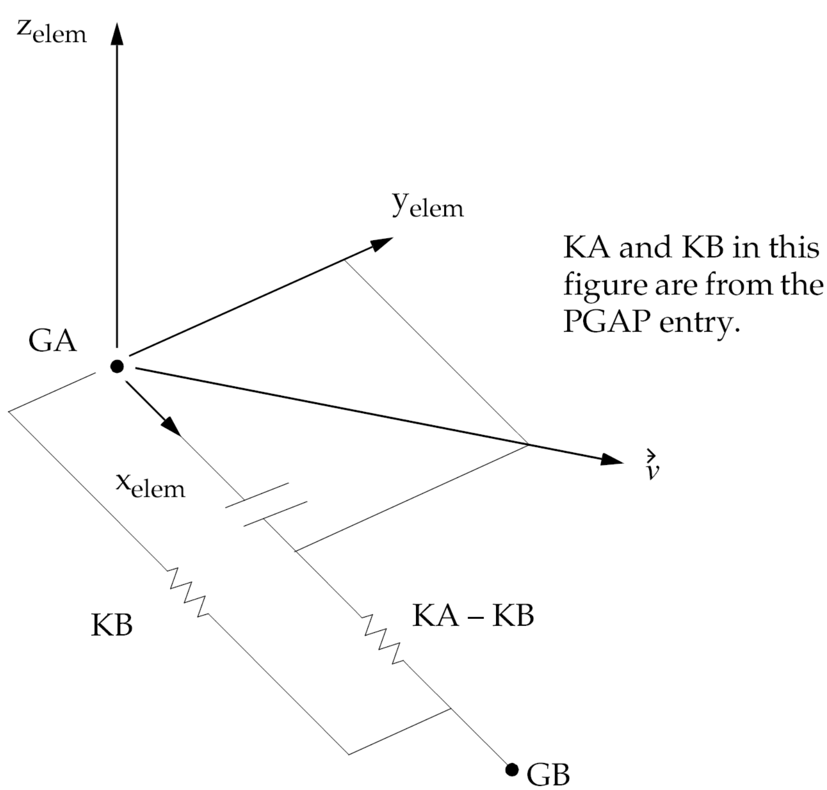

Figure 6-13. Gap Element Coordinate Systems

The properties for the gap elements are defined on the PGAP entry. The initial gap opening is defined by U_0 . If the gap is closed ($U_A - U_B \geq U_0$), the axial stiffness (KA) has a very large value (relative to the adjacent structure). When the gap is open, there is a small stiffness KB in the axial direction.

NX Nastran includes two types of gap elements: nonadaptive and adaptive. When you use the nonadaptive GAP element, you specify the anisotropic coefficients of friction (μ_1 and μ_2) for the frictional displacements. Also, the anisotropic coefficients of friction are replaced by the coefficients of static and dynamic friction μ_s and μ_k. On the PGAP continuation entry, the allowable penetration limit T_{max} should be specified because there is no default. In general, the recommended allowable penetration T_{max} is about 10% of the element thickness for plates or the equivalent thickness for other elements that are connected by GA and GB. When T_{max} is set to zero, the penalty values will not be adjusted adaptively.

Gap element forces (or stresses) and relative displacements are requested by the STRESS or FORCE Case Control command and computed in the element coordinate system. A positive axial force F_x indicates compression. For the element with friction, the magnitude of the slip displacement is always less than the shear displacement after the slip starts. For the element without friction, the shear displacements and slip displacements have the same value.

See Also

- “CGAP” in the *NX Nastran Quick Reference Guide*
- “Performing a 3-D Slide Line Contact Analysis” in the *NX Nastran Basic Nonlinear Analysis User’s Guide*

6.6 Concentrated Mass Elements (CONM1, CONM2)

You can use the concentrated mass elements to define a concentrated mass at a grid point. NX Nastran supports two forms of input of concentrated mass:

- CONM1
- CONM2

The CONM1 allows a general 6 × 6 symmetric mass matrix in a specified coordinate system to be assigned to a geometric grid point.

The CONM2 element allows a concentrated mass about its center of gravity to be specified. CONM2 lets you specify the offset of the center of gravity of the concentrated mass relative to grid point location, a reference coordinate system, the mass and a 3 × 3 symmetric matrix of mass moments of inertia measured from its center of gravity.

See Also

- “CONM1” in the *NX Nastran Quick Reference Guide*
- “CONM2” in the *NX Nastran Quick Reference Guide*

6.7 Hyperelastic Elements

The hyperelastic elements are intended for fully nonlinear (finite deformation) analysis including the effect of large strain and large rotation. Geometric nonlinearity is a subset of this type of analysis. In addition, the elements are especially designed to handle nonlinear elastic materials at the nearly incompressible limit. Volumetric locking avoidance is provided through a mixed formulation, based on a three field variational principle, with isoparametric displacement and discontinuous pressure and volumetric strain interpolations. Shear locking avoidance is provided through the use of second order elements.

You define the hyperelastic elements on the same connection entries as the other shell and solid elements. They are distinguished by their property entries. A PLPLANE or PLSOLID entry defines a hyperelastic element. The hyperelastic material, which is characterized by a generalized Rivlin polynomial form of order 5, applicable to compressible elastomers, is defined on the MATHP entry.

See Also

- "Elements for Nonlinear Analysis" in the *NX Nastran Basic Nonlinear Analysis User's Guide*

Hyperelastic Solid Elements

The following elements are available:

- CTETRA – Four-sided solid element with 4 to 10 nodes.
- CPENTA – Five-sided solid element with 6 to 15 nodes.
- CHEXA – Six-sided solid element with 8 to 20 nodes.

There's no element coordinate system associated with the hyperelastic solid elements. All output is in the basic coordinate system. The following quantities are output at the Gauss points:

- Cauchy stresses

$$\sigma_x,\ \sigma_y,\ \sigma_z,\ \tau_{xy},\ \tau_{yz},\ \tau_{zx}$$

- Pressure

$$p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$$

- Logarithmic strains

$$\varepsilon = \sum_{l=1}^{3} \ln\gamma_l N_l N_l^T$$

- Volumetric strain

$$J - 1 = \frac{dV - dV_0}{dV_0}$$

See Also

- "Solid Elements (CTETRA, CPENTA, CHEXA)"

Hyperelastic Plane Elements

These are plane strain elements defined on the following connectivities:

- CQUAD – Quadrilateral element with 4 to 9 nodes. When the center node is missing, this element may also be specified on a CQUAD8 connectivity entry. When all edge nodes are missing, the CQUAD4 connectivity may be used.
- CTRIA3 – Triangular element with 3 nodes.
- CTRIA6 – Triangular element with 3 to 6 nodes.

Figure 6-14 shows the element connectivity for the CQUAD element.

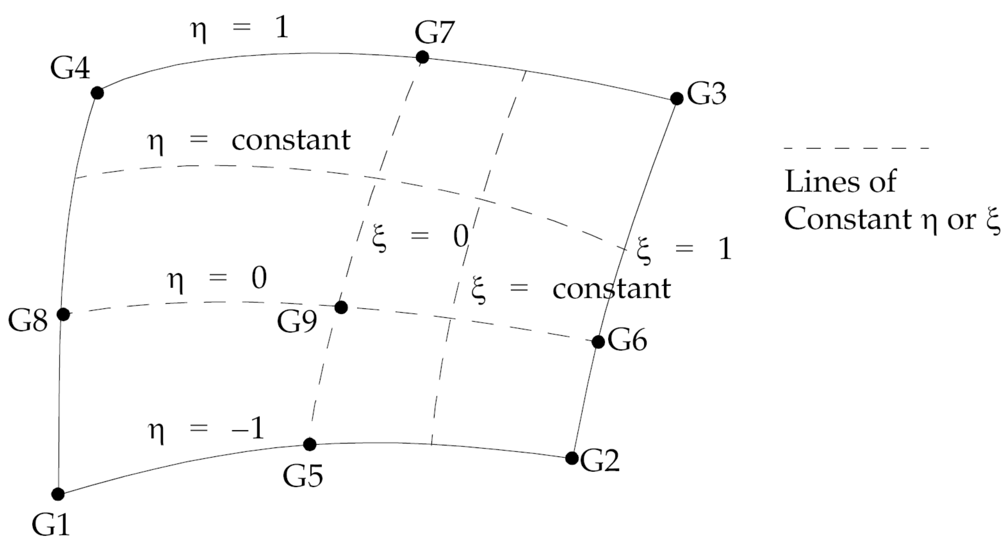

Figure 6-14. CQUAD Element

Note, however, that there is no element coordinate system associated with the hyperelastic plane elements. All output is in the CID coordinate system. Cauchy stresses σ_x, σ_y, σ_z, τ_{xy}, pressure $p = 1/3(\sigma_x + \sigma_y + \sigma_z)$, logarithmic strains and volumetric strain are output at the Gauss points.

The plane of deformation is the XY plane of the CID coordinate system, defined on the PLPLANE property entry. The model and all loading must lie on this plane, which, by default, is the XY plane of the basic coordinate system. The displacement along the Z axis of the CID coordinate system is zero or constant.

Hyperelastic Axisymmetric Elements

Hyperelasticity can be modeled with the following axisymmetric elements:

- CTRAX3 – Triangular axisymmetric element with 3 nodes.
- CTRAX6 – Triangular axisymmetric element with 6 nodes.
- CQUADX4 – Quadrilateral axisymmetric element with 4 nodes.
- CQUADX8 – Quadrilateral axisymmetric element with 8 nodes.

When using these elements, you must construct the model such that the elements and loading lie in one of the following planes:

- The XY-plane of the basic coordinate system with the Y-axis as the axisymmetric axis.
- the XZ-plane of the basic coordinate system with the Z-axis as the axisymmetric axis.

Pressure loads with follower force characteristics may be applied to the edges of axisymmetric elements with PLOADX1 entries.

Temperature loads may be specified for all hyperelastic elements on the TEMP and TEMD entries. The hyperelastic material, however, may not be temperature dependent. Temperature affects the stress-strain relation.

GPSTRESS and FORCE (or ELFORCE) output is not available for hyperelastic elements.

6.8 Interface Elements

The interface elements are primarily used when performing global local analyses.

The interface elements use a hybrid variational formulation with Lagrange multipliers, developed by NASA Langley Research Center. There are displacement variables defined on the interface element, in order to avoid making the interface too stiff, such as a rigid element. There are also Lagrange multipliers defined on each boundary, which represent the forces between the boundaries and the interface element. This formulation is energy-based and results in a compliant interface.

Curve Interface Elements

Interface elements allow you to connect dissimilar meshes over a common geometric boundary, instead of using transition meshes or constraint conditions. Primary applications where you might specify the interface elements manually include: facilitating global-local analysis, where a patch of elements may be removed from the global model and replaced by a denser patch for a local detail, without having to transition to the surrounding area; and connecting meshes built by different engineering organizations, such as a wing to the fuselage of an airplane.

Primary applications where the interface elements could be generated automatically are related to automeshers, which may be required to transition between large and small elements between mesh regions; and h-refinement, where subdivided elements may be adjacent to undivided elements without a transition area.

Dissimilar meshes can occur with global-local analysis, where part of the structure is modeled as the area of primary interest in which detailed stress distributions are required, and part of the structure is modeled as the area of secondary interest through which load paths are passed into the area of primary interest.

Generally the area of primary interest has a finer mesh than the area of secondary interest and, therefore, a transition area is required. Severe transitions generally produce elements that are heavily distorted, which can result in poor stresses and poor load transfer into the area of primary interest. An example of using interface elements to avoid such transitions is shown in Figure 6-15. Similarly, a patch of elements may be removed from the global model and replaced by a denser patch for local detail.

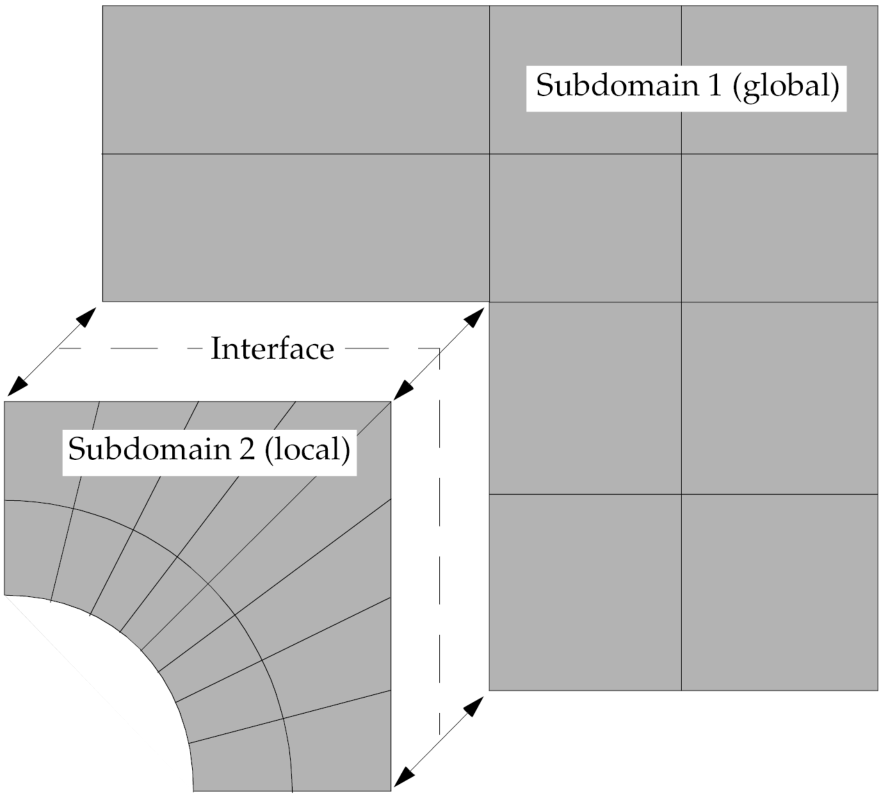

Figure 6-15. Example of Interface Elements (Exploded View)

In large system assemblies, different analysts or even different organizations may have created different components of the model, such as the wing and the fuselage of an airplane. Unless they have carefully coordinated their efforts, the finite element meshes of the different components may not match at the interfaces.

Dissimilar meshes generated by the analysis program can also arise with automeshers, which may be required to transition between large elements and small elements in a small area. Many automeshers generate tetrahedral meshes for solids, and distorted tetrahedra may be more susceptible to poor results. Interface elements are particularly useful when a transition is needed between large and small elements within a small area.

When mesh refinement is performed, subdivided elements may be adjacent to undivided elements with no room for a transition area. Without some kind of interface element, the subdivision would need to be carried out to the model boundary or otherwise transitioned out.

It is important to note that the interface elements provide a tool for connecting dissimilar meshes, but they do not increase the accuracy of the mesh. As with any interface formulation, the hybrid variational technology, which imposes continuity conditions in a weak form, can not increase the accuracy of the adjacent subdomains. For instance, if a single element edge on one boundary is connected to many element edges on the other boundary, the analysis is going to be limited to the accuracy of the boundary containing the single element edge. This restriction should be considered when deciding how close to the areas of primary interest to put the interface elements.