

Contact Modelling in Femap with NX Nastran

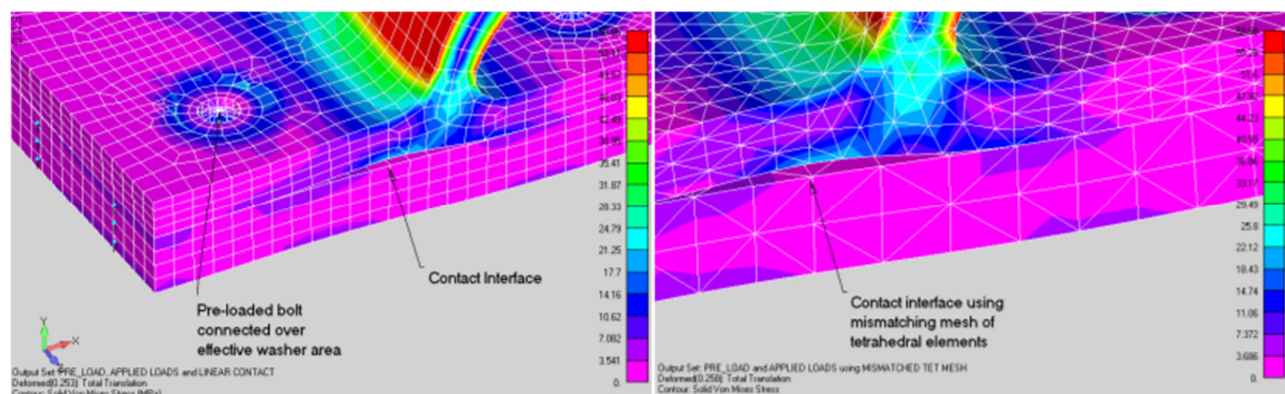
Contact can be divided into three styles:

- linear contact
- non-linear gap / slide-line contact
- general 3D contact

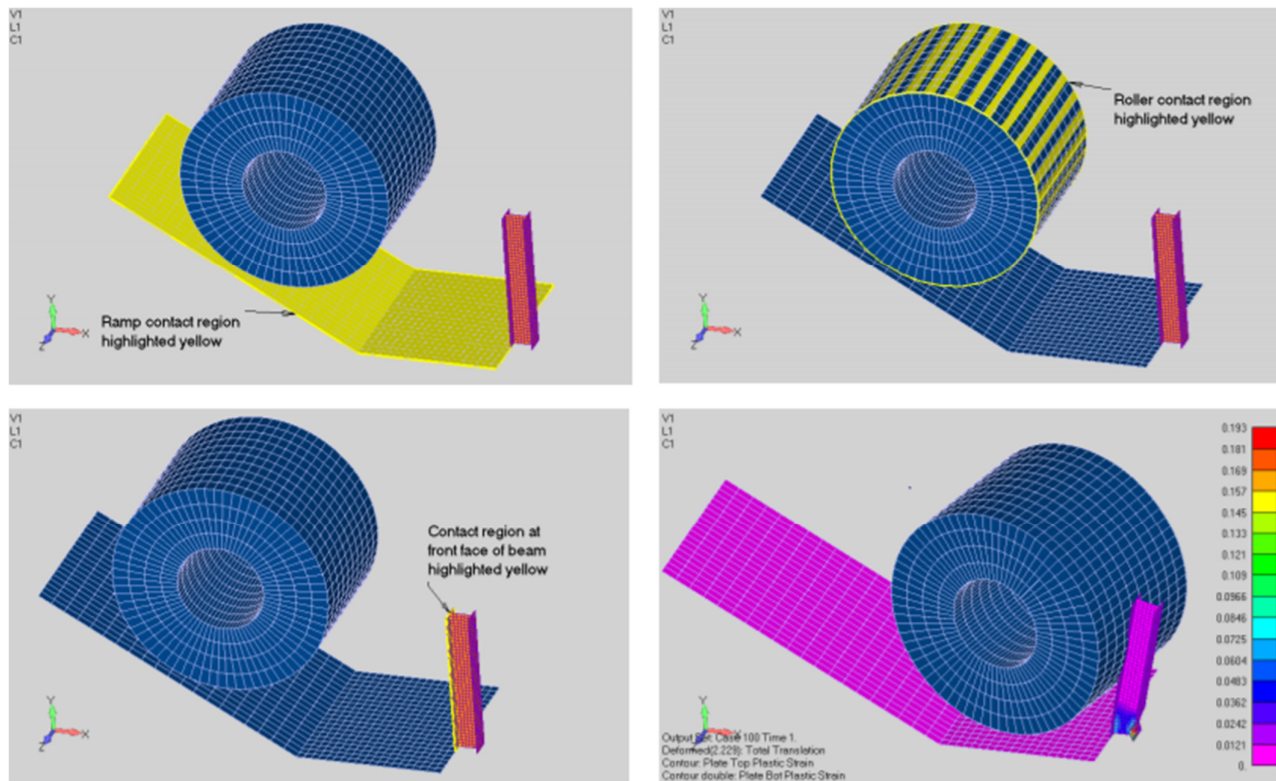
This document will briefly describe linear and general 3D contact.

Standard in Femap with NX Nastran is the ability to perform "linear contact" with similar or dissimilar plate or solid mesh. The most accurate results are produced if the mesh matches; however there are many situations where matching the mesh is either un-necessary, difficult or impossible within the confines of practical computing resources.

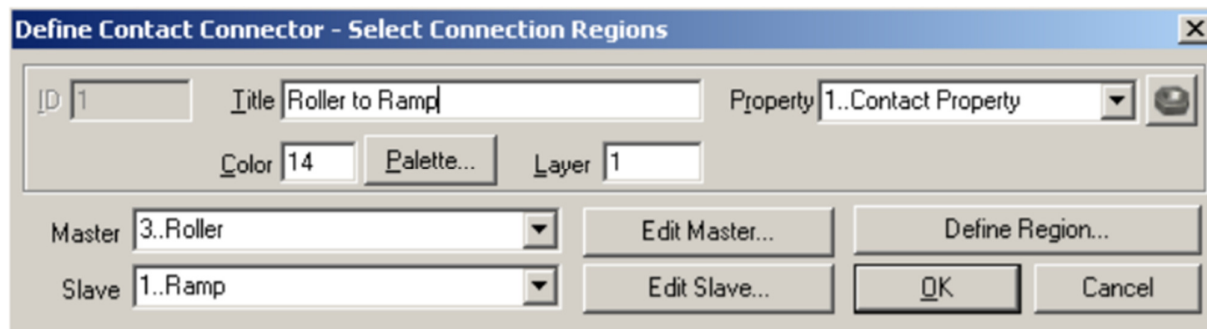
For those with Nastran experience, linear contact makes the modelling of "small-motion" contact interfaces much easier. This includes interfaces which use (for example) bolts, rivets or tie rods. Preload within the fastener is also readily modelled if desired. Note that a small motion interface is one where the contact calculation determines whether the adjacent areas across the contact interfaces are separated or closed (stuck or sliding, if friction is included) under the applied loading conditions.



On the other hand, general 3D contact is where the contact interfaces move "significantly" relative to each other, and is a characteristic of Femap with NX Nastran Advanced Non-Linear. However, the process of setting up the contact is almost identical. The key components of modelling contact are "regions", "connectors" and "connection properties". In the classic logical Nastran structure that engenders efficiency and ease-of-modification: connectors reference regions and contact properties. A region defines one/more geometric surfaces (or collection of elements) which may potentially come in contact with other region(s).

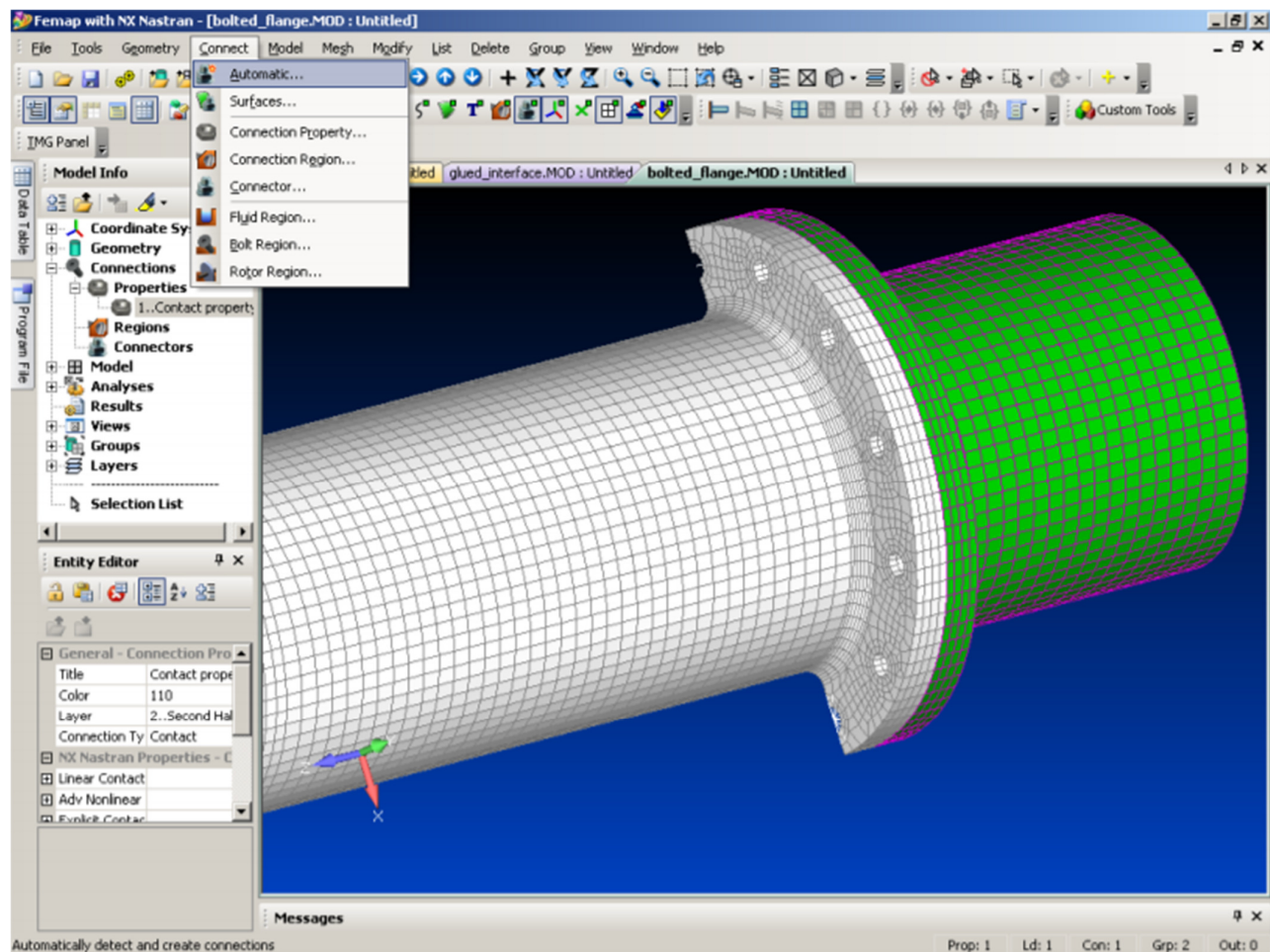


A connector links pairs of regions which can potentially interact. For example, creating the contact connector between the roller and ramp above is done using the simple method below.



A connection property describes physical characteristics of the contact interface, such as friction coefficient. The user can choose whether or not to account for plate thickness in establishing contact (for connections involving plate elements), and, among other controls, whether "continuous normals" are used to smooth out a faceted finite element representation in general 3D contact. Other parameters of a contact property control the mathematical behaviour of the interface in open and closed condition and, for most contact problems, can remain as their default values.

The easiest way to model contact in a solid assembly is to use Connect | Automatic. In this method, all the solids which will potentially interact are chosen, then simply choose whether the interface should be a "glue" or "contact" interface. Femap automatically creates the regions, connectors and contact properties.

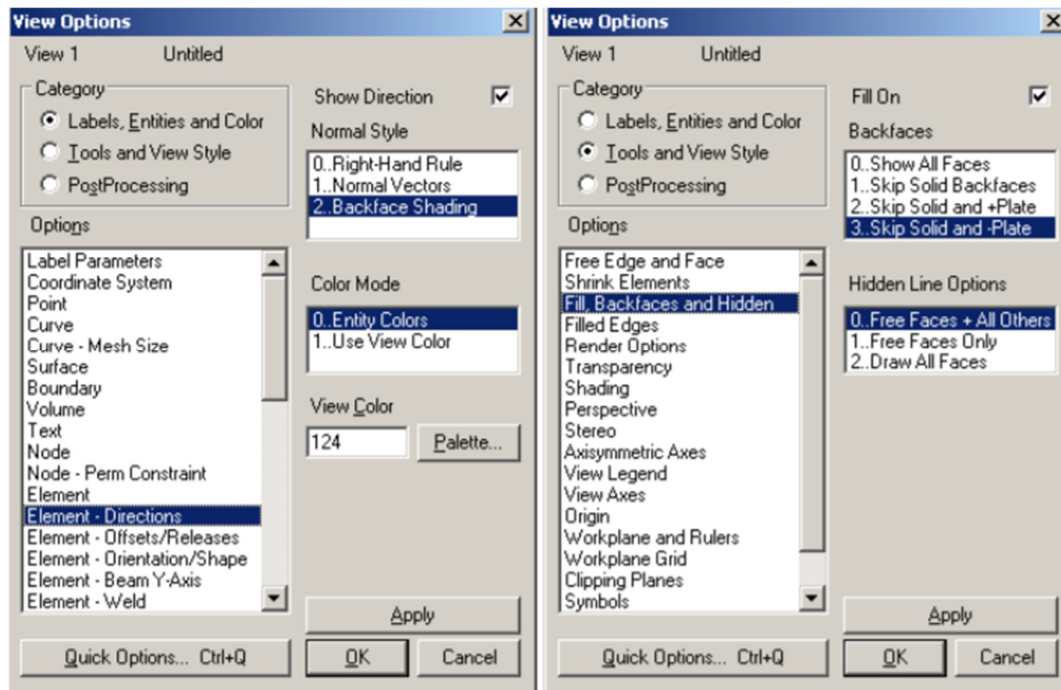


Otherwise, the user can easily create contact interactions manually:

1. Create regions using geometry surfaces or elements to define the potential contacting areas;
2. Create a contact property to represent the behaviour of the interface (eg. glued or contact, and friction if needed);
3. Create connectors. Each connector is simply two contact regions and a contact property. Unless the friction coefficient is different for different interfaces, a model with multiple connectors may need only one connection property.

Tips for getting the best results from contact modelling:

1. Best analysis results are achieved if the mesh sizing across a contact interface is similar. If not, then better results are achieved if the finer meshed region is the "Master". This is because contact relationships are established from the master to the slave – so a higher resolution of contact definition is created if the finer mesh area is defined as the master region.
2. When creating contact regions which relate to plate elements, you need to know which side is the "front" and "back" face of the surface. In Femap it is easy to visualise this by only displaying the front face of a surface. Or by shading the "backface" of elements. This is done using the common View | Options menu (F6).

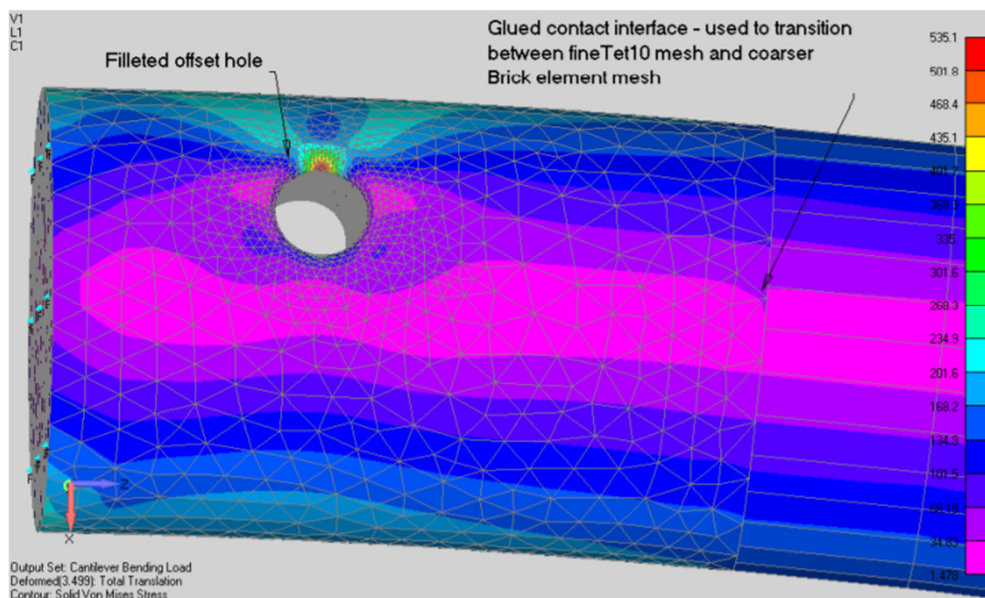


When the region is defined, you can choose either the positive or negative side of any surfaces in the contact region to be used as the contact side (in Advanced Non-Linear, double sided contact is possible).

3. Do not merge nodes across a contact interface!

4. Even in linear analysis, contact models take longer to run, as the problem iterates around the closed/open status of the contact segments. The time for analysis will increase if there are hundreds or thousands of active contact segments. The runtime will not increase by much if there are a large number of contact segments defined, but only a small number are active.

5. A glued interface is useful in areas where large transitions in mesh refinement are required. It is best to create the glued interface at a reasonable distance from a detailed area where precise stresses are required, as there will be some degree of stress discontinuity across an interface which has widely varying mesh densities. A glue interface can be set when defining a contact property.



A glued interface is also useful across interfaces where peeling of the interface is known to be minimal or non-existent, as glued interfaces analyse faster than contact interfaces.