Overcoming Model Singularities in Nastran (or any other finite element system)

Model singularities are the most common error messages for which we receive support questions, and can be a source of frustration for many inexperienced (and experienced) analysts. Here are some troubleshooting tips for curing this common modelling error.

In NX Nastran, the symptom of this problem is User Fatal Message 9137 (previously User Fatal Message 9050 in MSC Nastran), "Run terminated due to excessive pivot ratios". Note that Nastran never reports this error through any randomness or unreliability - if you receive this message, then your representation of the model physics needs to be investigated to overcome the problem.

Note that this discussion relates to Linear Static Analysis, for which you are attempting to find a single unique solution to a glorified version of F=kx. If a unique solution is not possible due to a modelling error, this "singularity" will cause User Fatal Message 9137.

Modelling oversight in order of likelihood:

The term "rigid body motion" is commonly used in finite element analysis. Rigid body motion means the structure is "floating" without any resulting stress/strain. This means some or all of your modelled part / structure / assembly is insufficiently connected to "the ground".

Note that having satisfactory global constraints is absolutely independent of the applied load. Thus, even though your structure may only have vertical loads, there must be some form of horizontal constraint for Nastran to find one unique solution.

There are a minimum set of constraints which are required to stop the structure floating in all of the potential 6 global directions ("degrees of freedom"), namely: translation in X (TX), translation in Y (TY), translation in Z (TZ) and rotations about these 3 axes (RX, RY, RZ). Note that even though you may choose a location and "fix" it completely (ie. in all 3 translations and all 3 rotations), it is important to understand whether the point you have chosen has stiffness in all 6 directions.

As an example, each node in a typical solid element only supports the 3 translational degrees of freedom, so a pinned constraint (TX, TY, TZ) and a fixed constraint (TX, TY, TZ, RX, RY, RZ) are identical when applied to a node on a solid element. Therefore, constraints must be applied to at least 3 separate non colinear locations in a 3D all-solid model in order to prevent some type rigid body motion.

Similarly, nodes on a typical plate element (there are exceptions) have stiffness in only 5 of the 6 possible directions - a node on a plate element does not have stiffness in the axis normal to the plane of the element, so a plate model with a single node fixed in all 6 directions can still spin freely about the normal to the plate element (and thus produce a USER FATAL 9137).

A structure supported by eg. a bed of vertical axial springs will also report UFM 9137 even if the bottom of all the springs are fully fixed. This is because axial springs do not have any bending or shear stiffness. The structure will be supported vertically and in "pitch" and "roll", but direct fore/aft and side/side and yaw motion will not be supported by axial springs. Again, it is important to re-emphasise that the error will occur irrespective of the type/direction of applied loads.

> **Solution: Inspect all the constraints in your structure to understand what they are connected to and which constraint(s) at what location(s) are supporting the motion in all the six potential directions of motion.**

It is critical to note that just because you apply enough constraints to prevent UFM 9137, this does not mean you automatically have the "right answer". We have observed dozens of situations where structures have been over-constrained "just to get results", but the constraints have not been good representations of the physical behaviour.

A subsection of the model has insufficient connectivity to a satisfactorily constrained structure.

This is actually a more common error than Point 1 above, but it is easier to describe these situations once global constraints are fully understood.

The primary issue here is that not only does the full structure need a satisfactory set of global constraints, but every element in the model must have enough support/connection to prevent floating rigid body motion.

The most common error is parts of the model being completely disconnected from the parts which might have proper global constraints. Note that "being connected" typically means "sharing one or more nodes".

So, poor connectivity may be a simple issue of nodes not being merged, or a slightly more complicated issue of mesh sizing being different across different parts of the structure. If the meshes do not match up, then (even though the mesh might be close/adjacent) these parts of the structure are not connected properly.

> **Solution: In Femap, use Tools | Check | Coincident Nodes to merge coincident nodes and/or use F5 -> Free Edge as the Model Style to inspect for "cracks" in plate and solid models to show if the mesh does not match up**

If the mesh does not match up, part of the mesh typically needs to be deleted and remeshed with a matching mesh sizing.

If it is difficult to identify the problem area, copy the list of reported nodes from the Analysis Monitor (or the Nastran .f06 file), then in Femap, use Window | Show Entity -> Nodes -> (Pick -> Paste in Entity Selection Dialog).

Another method is to run a natural frequency analysis and then animate the "close to 0.0 Hz" modes. These modes represent the way in which parts of (or the whole) structure can "float" unsupported. If you use natural frequency analysis to diagnose static analysis constraint or connectivity problems, it is essential that your materials include realistic densities

The stiffness of the connecting elements do not support certain types of motion.

For example, if a straight edge of a plate element mesh is connected to a line of nodes on solid elements, then this connection is a hinge.

A hinge is produced because the nodes of solid elements only have stiffness in the translational directions, but not rotation. Thus even if there are sufficient global constraints for the model, the hinge joint may allow some part of the model to "flap" freely.

Note that if it were the end of a tube of plate elements connecting to a circle of nodes on solid elements, there would be no rigid body motion, however, the load transfer across the plate-tosolid transition may not be satisfactory modelling, without some additional effort (not covered in this tech tip).

Similarly a beam element connected to a node on a solid element can pivot and swivel just like a ball joint.

So it is important to understand what degrees of freedom are supported by (ie. what directions of stiffness are present) in each type of element you are using in your model. As a quick, rough guide:
- Solids: TX, TY, TZ (ie. translational stiffness only)
- Plate/Shells: TX, TY, TZ, RX, RY. Note that these directions are with respect to the local coordinates of the element. Thus, TX and TY are in the plane of the element, and RZ (no stiffness supported) is commonly referred to as the "drilling" degree of freedom.
- Beams: TX,TY,TZ, RX, RY, RZ (all degrees of freedom).

These directions are also with respect to the local coordinates of the element. Thus TX is the axial direction of the beam. TY and TZ are in the local section directions of the beam. Note that you can switch off individual stiffnesses in beams via "pin releases" to represent ball joints, hinges and sliders - but you must make sure that you do not inadvertently create floating rigid body mechanisms (thus UFM 9137).

Springs: TX only - typically in the line formed between the two nodes which define the spring. Thus, if using springs to connect two parts of a structure, it is important to understand how all of the 6 degrees of freedom are supported in each section of the structure.

> **Solution: It can be more difficult to identify this problem, as the floating characteristics can be quite subtle. However, highlighting listed nodes and running normal modes analysis as described in Point 2 above are the best methods.**

Enormous differences in stiffness can cause singularities (thus UFM 9137)

This can also be quite a subtle problem, which arises if unsuitable values are used for section sizes, thickness or material stiffnesses.

As an example, let's say you wish to connect two structures together, and are unfamiliar with the use of rigid elements. Say you are working in SI, and you create a beam of area 100, I11 =100 and I22 = 100. Note that this is a beam which is the equivalent of a 10m x 10m solid block. This will be many orders of magnitude stiffer than the surrounding structure (for most models!) and may fall over the threshold "maxratio" (and thus produce UFM 9137). This is because the mathematical solution can start losing precision if there are enormous variations of stiffness within the model.

A similar problem can occur if modelling a "stiff" structure and then using "watch springs" to add some stiffness where your structure might otherwise be floating.

> **Solution: Once again, highlight some or all of the listed problem nodes in Femap to understand where the problem area(s) lie. Use sensible values of stiffness for linkages and "soft" constraint springs which produce the desired outcome within useful limits of engineering precision. Do not choose extreme values in these cases. Note that the MAXRATIO threshold can be changed, but it is better to find the cause of the error within the model**

Contact interactions

Note that this section can be applicable to non-linear static analysis as well as linear static analysis.

It is important to understand that if a model requires contact to be established before a unique solution can be found, then model singularities will occur if the contacting structure can "float" or "slide" (ie. exhibit rigid body motion) prior to the contact being established.

In a non-linear analysis, the symptom can be failure to achieve a converged solution (even if the load step is reduced to a very small value) because no equilibrium can be established until the contact occurs. In linear static analysis (SOL 101 or SESTATICS in Nastran) the symptom is the standard singularity error User Fatal Message 9137.

Solution: If it is too inconvenient to get everything into initial contact, then "soft springs" can be used to provide enough connection between "fixed" and "moving" part(s) of the structure.

The springs should be soft enough to make negligible difference to the results. However, noting the problems identified in Item 4 above, the springs should not be too many orders of magnitude softer than the surrounding structure.

Note that as you become more experienced, it is possible to adjust contact properties to create the effect of contact interactions in advance of the contact actually occurring. Note that the inclusion of contact friction can remove some degrees of singularity from a contacting structure. Similarly, it is important to note that if friction is zero, then sideways motion at a contact interface MUST be supported in some other way on the "floating" part of the structure, otherwise a singularity error will also occur. Again, it is important to note that this requirement is independent of the direction of any applied loads.

These brief guidelines have been created for Nastran users; however the same principle applies to other analysis codes, such as Abaqus, Ansys, Marc, Cosmos, Algor, Strand and any others.