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

EnDuraSim has decades of hands on experience in FEA, particularly using Femap and NX Nastran.
We provide general training in FEA or specific training for Femap and NX Nastran.
We provide quality FEA consulting, typically fixed price contract for defined scope of work.
We provide technical support for *Femap with NX Nastran*.
We provide Femap API programming products and services to enhance analyst productivity.
Contact us for further information.

Model singularities are still the most common source of FEA analysis error.
Here are some troubleshooting tips for curing this common modelling error. The principles apply to virtually any FEA package, but this guide is most helpful to Nastran and Femap users.

In NX Nastran, the problem is indicated by "User Fatal Message 9137 (SEKRRS)"; previously User Fatal Message 9050 in MSC Nastran), "Run terminated due to excessive pivot ratios in Matrix KLL". It always represents a modelling error or FEA oversight.

The discussion mainly relates to Linear Static Analysis, which is to find a unique solution to a matrix (simultaneous equation) version of F=kx. If a unique solution is not possible due to a singularity modelling error, it will cause User Fatal Message 9137 in Nastran.

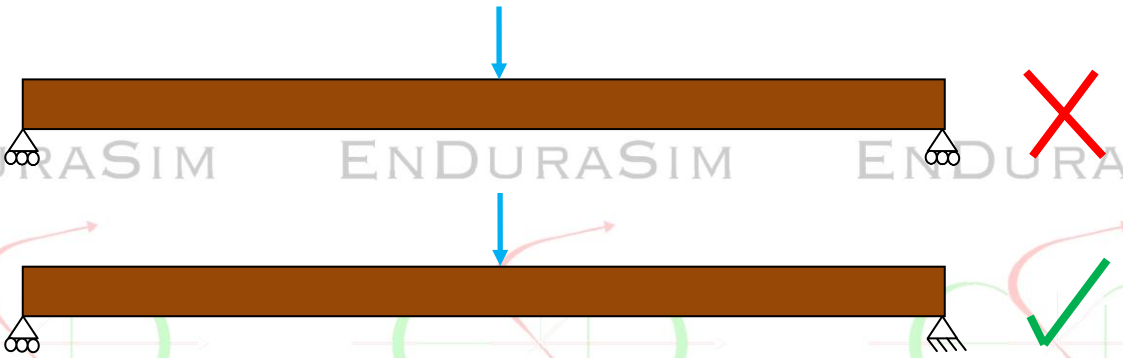
(*** See footnote at the end of this document which explains the difference between the "Grid Point Singularity Table" and the error information from UFM 9137).

Modelling problems in approximate order of likelihood:

1. The model has insufficient global constraints.

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

Note that having satisfactory global constraints is independent of the applied load (applied loads affect contact models - technically a non-linearity - discussed briefly later). Thus, even though a structure may only have vertical loads, it must include a horizontal constraint for Nastran to mathematically have a "reference zero" and find a unique solution.



The first case above has insufficient global constraints (floats horizontally) hence fails analysis.

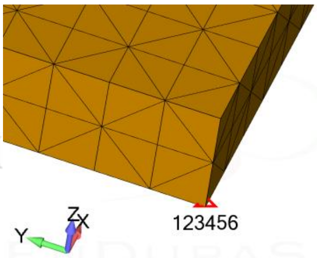
It is critical to note that sufficient constraint (to avoid a singularity error) never automatically implies correct constraint. There are innumerable ways in which FEA models can run and produce results, but still be wrong if not modelled and analysed properly.

To avoid singularity error, 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:

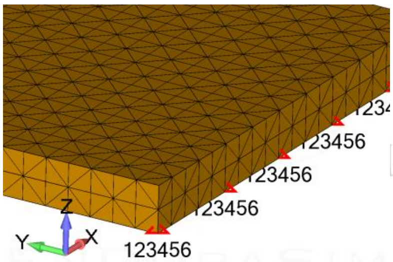
Translation in X (T1), Translation in Y (T2), Translation in Z (T3); and
Rotations about these 3 axes (R1, R2, R3).

Although a point (node or grid) is "fixed" completely (ie. in all 3 translations and all 3 rotations), it is important to consider whether that node has stiffness in all its 6 directions (DoFs 1,2,3,4,5,6 in the basic global coord system direction, by default).

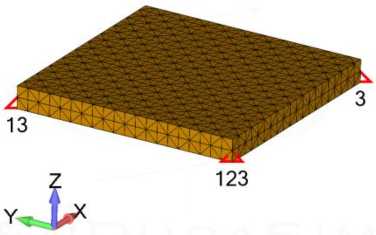
For example, each node in a typical FEA solid element only supports the 3 translational degrees of freedom, so a pinned constraint (TX, TY, TZ, ie. 123) and a fixed constraint (TX, TY, TZ, RX, RY, RZ, ie. 123456) are identical when applied to a node on a solid element.



This constraint alone = ball joint, as solid element nodes have no RX, RY, RZ stiffness. Thus, there is a singularity error. Direction of load is irrelevant.



These constraints alone = hinge, as solid element nodes have no RX, RY, RZ stiffness. Thus, there is a singularity error. Direction of load is irrelevant.



This is *one example* of the minimum constraints an all-solid FEA model needs to avoid a global singularity error.

Therefore, constraints must be applied to at least 3 separate non co-linear locations in a 3D all-solid model in order to prevent some type of global rigid body motion. If the only constrained locations of a solid model are co-linear, then the structure can rotate about the line of the constrained locations.

Similarly, nodes on a typical plate element (there are exceptions) have stiffness in only 5 of the 6 possible directions - a plate element node does not have stiffness in the axis normal to the plane of the element ("drilling" DoF). Thus, a ***flat*** plate model with a single node fixed in all 6 directions can still spin freely about the normal to the plate element (and thus UFM 9137 singularity).

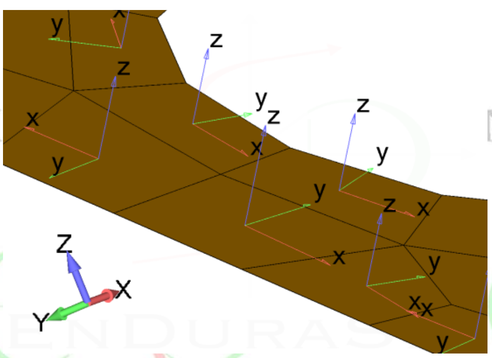
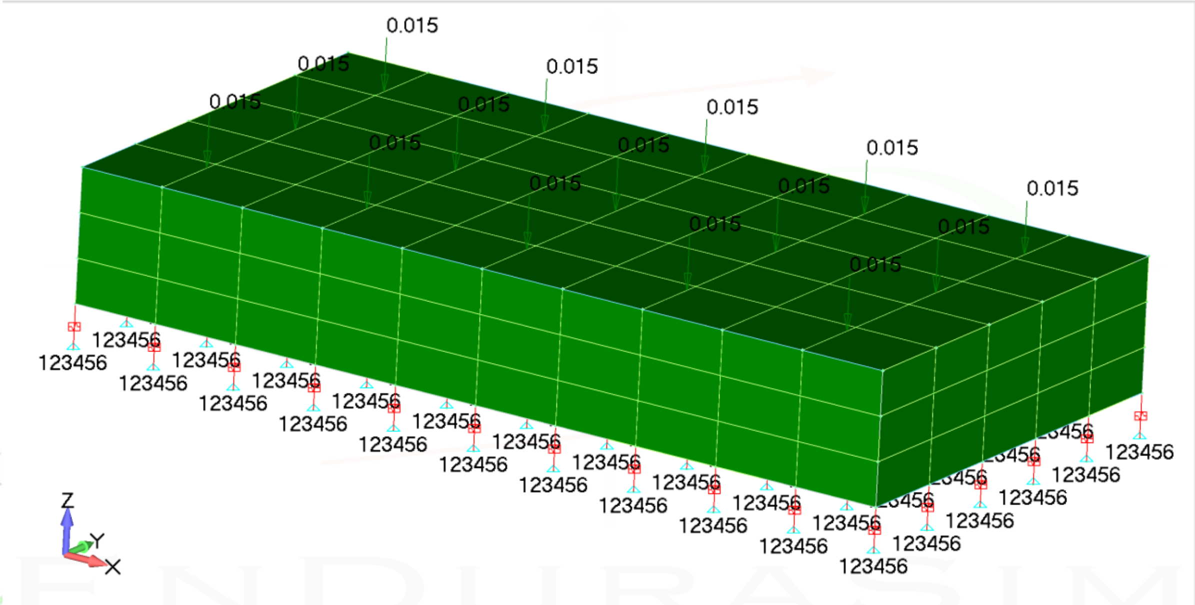


Plate Finite Element Mesh with element coordinates displayed. Plate element nodes (generally) do not have rotation stiffness around the local element Z direction, ie. no stiffness in *local* RZ.

Thus a single node constrained in all DoF (default = global) of this flat plate mesh does not prevent the mesh from spinning around the *local RZ* direction of the plane of elements.

Note that if a plate mesh is *not flat*, bending stiffness from one element contributes a component of stiffness to the otherwise zero local RZ (drilling) stiffness in an adjacent element at a non-flat junction. One fully fixed node at such a junction would avoid a singularity error (although it may well be a poor FEA model).

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 simple FEA axial springs do not have any bending or shear stiffness (in contrast to other 1D elements like BEAMS or CBUSHES). 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.



This simple model appears to be very well constrained, but will produce a singularity error because the supporting axial springs only have stiffness in the global TZ direction. The FEA mesh is thus not properly supported in the TX, TY and RZ directions (1,2 and 6 global DoF). The presence of only vertical load is irrelevant to the singularity problem.

Solution:

Inspect all the constraints in the structure to understand what they are connected to and which constraint(s) at what location(s) are preventing rigid body motion in all the six potential directions of global motion.

Again, note that even if singularities are resolved, this does not mean the result is correct. We have observed dozens of situations where structures were over-constrained "just to get results", but the constraints are poor representations of the physical behaviour.

Also many solvers have "fudge solutions" for singularity (in Nastran this is called BAILOUT which can be activated in Femap via the Analysis Manager -> Bulk Data Options of the Analysis Set). These methods must *only ever* be used to help diagnose the *modelling problem*. The results produced from a BAILOUT solution are almost certainly wrong for the *intended* physics.

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

This is probably more common than Point 1 above, but it is easier to describe these situations once global constraints are fully understood. Not only does the full structure need a satisfactory set of global constraints, but *every* element in the model must also have enough support/connection to prevent floating rigid body (un-resisted) motion of each element.

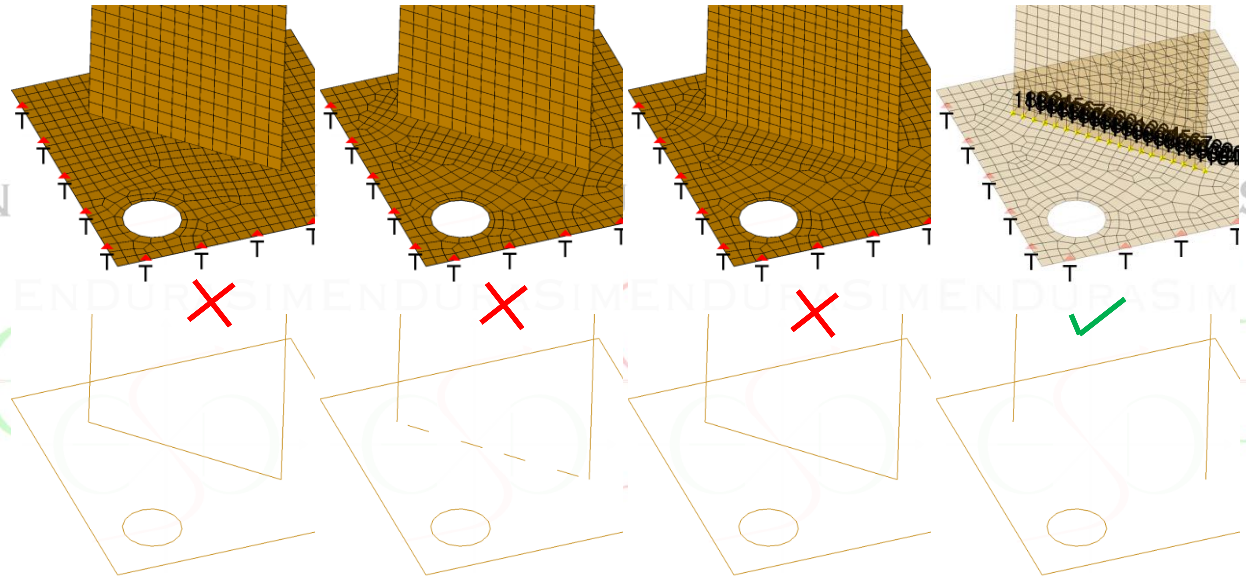
The most common problem is when part(s) of the model is/are completely disconnected from the part(s) which might have proper global constraints. Note that elements "being connected" typically means "sharing one or more nodes". So, poor connectivity may be a simple issue of

coincident nodes from adjacent parts not being merged, or the more complicated issue of mesh sizing being mismatched across different parts which are supposed to be joined. 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. Again, note that intermittent connection may help solve singularity errors, but not produce satisfactory results.

Solution:

For Femap analysts, 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 sizing is mismatched where parts are supposed to be joined, then part of the mesh typically needs to be deleted and re-meshed with a matching mesh size so the coincident nodes can be merged (= elements connected properly).

The images below show three common mesh connection problems and the final resolution.



In this case the geometry has not been edited correctly, and the vertical mesh has no connection with the horizontal mesh. The Free Edge view (lower image) shows there is no connection. The global constraints are adequate, but the model *will* produce a singularity error.

In this case the geometry has been edited correctly, but the mesh sizing is mismatched at the junction. The Free Edge view (lower image) shows intermittent connection after merging nodes with a larger tolerance. The model *will not* produce a singularity, but unless stitch welded in reality, the model is *wrong*.

The mesh looks correct here, but the Free Edge view (lower image) shows the nodes on the vertical mesh are not merged with the nodes on the horizontal mesh. No connection means the vertical mesh can "float". The model *will* produce a singularity error.

The mesh is matched and with the highlighted nodes merged, the junction connection between vertical and horizontal mesh is complete. The model will not produce a singularity error.

If it is difficult to identify the problem area, copy the list of reported nodes (in Nastran, the list is immediately prior to UFM 9137) from the Analysis Monitor (easier via the Nastran .f06 text file), then in Femap, use Window | Show Entity -> Nodes -> (Pick -> Paste in Entity Selection Dialog).

<u>Another method is to run natural frequency analysis</u> (using the same constraints as the static analysis) and animate the modes which are "very close to 0.0 Hz". These modes show the way in which unconnected parts of (or the whole) structure can "float" unsupported. Using natural frequency analysis to diagnose static analysis constraint or connectivity problems REQUIRES

realistic material densities. There will be 6 x zero Hz modes for each independent section of the structure which is wholly unconnected to (another part of the structure which has) proper constraints. For additional clarity, the example image from Section 1 above would exhibit 3 x zero Hz modes: one for each direction in which the structure could float.

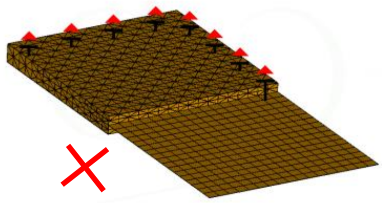
Also, see our Free Femap API page for some faster tools to help model checking. These include API's which highlight orphan plate and solid mesh (mesh which is no longer associated with geometry). Such mesh is usually an unintended by-product of a modelling error, and is often not connected properly to the structure. Another API highlights recursively connected elements. For complex assemblies this can help determine whether meshed parts or sub-assemblies are not connected.

EnDuraSim can provide training in API programming and provide custom-written APIs to automate model-checking and results processing - thus saving time and reducing errors in complex models / procedures.

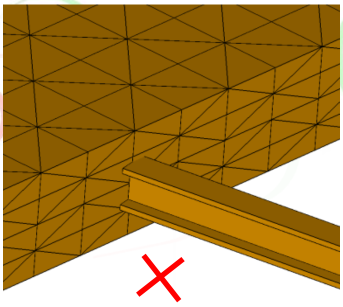
3. The stiffness of the connecting elements do not resist certain types of motion.

This was briefly introduced in the section on Global Constraints.

For example, if a straight edge of a plate element mesh is properly 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.



This simple model has sufficient global constraints. The nodes are merged at the junction of the plate and solid mesh, so it is "connected". However a singularity error *will* occur, because the solid element nodes have no rotation stiffness. Thus the junction is a hinge and the plate elements can flap freely about the line of the hinge. Any vertical constraint on the plate mesh away from the hinge will resolve the singularity (but is unlikely to make the model correct!)



This beam element connected to a node on the solid mesh is a ball joint. In the absence of sufficient support for the beam (eg. constraints; or other properly supported mesh; or distributing the point connection, eg. via rigid elements) this connection *will* cause a singularity error.

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 hinge motion, however, the load transfer across the plate-to-solid transition may not be satisfactory modelling, without some additional effort (one method is explained in our Tech Tip on using glue to transition from plate to solid mesh).

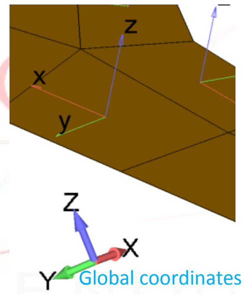
Similarly, a beam element connected to a node on a solid element can pivot and swivel just like a ball joint.

Thus, it is important to understand what degrees of freedom are supported (ie. what directions of stiffness are present at the nodes) in each type of element in the model. As a quick, rough guide:

Solids: TX, TY, TZ (ie. translational stiffness only).

Plate/Shells: TX, TY, TZ, RX, RY.

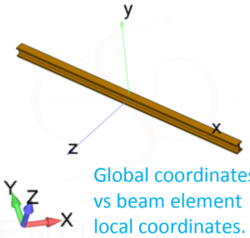
Note that these directions are with respect to the local coordinates of the plate element. Thus, TX and TY are in the plane of the element, and RZ (no stiffness supported) is commonly referred to as the "drilling" degree of freedom.



Global coordinates vs plate element local coordinates.

Beams: TX,TY,TZ, RX, RY, RZ (all degrees of freedom).

These directions are also with respect to the local coordinates of the beam element. TX is the axial direction of the beam. TY and TZ are in the local section directions of the beam. Individual stiffnesses can be switched off in beams via "pin releases" to represent ball joints, hinges and sliders - but avoid inadvertently creating floating/swivelling rigid body mechanisms (thus singularity error UFM 9137).



Global coordinates vs beam element local coordinates.

Axial Springs: TX only - typically in the direction 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, particularly if the floating section(s) is/are quite small. However, highlighting listed singular nodes (in the .f06 file in Nastran) will help to see if the problem belongs to a specific local area. Otherwise, running normal modes analysis as described in Point 2 above is also useful, so that the zero Hz modes of the floating part(s) can be animated.

4. 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 the analyst wishes to connect two structures together, but is unfamiliar with the use of rigid elements. Working in standard SI (m,kg,sec) , the analyst creates a beam of area 100, I11 =1000 and I22 = 1000. This is a beam which is roughly equivalent to a 10m x 10m square section. 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 to {F}=[K]{x} starts 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 nominal support where the structure might otherwise be floating (eg. to prevent floating sub-parts in Contact models – see Section 5 below).

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, and use *reasonable* large or small values.

5. Contact interactions. This section is also relevant to non-linear static analysis.

Contact often adds extra potential for singularity problems. Nastran can include contact in the "linear" static solution, but technically any contact is a form of non-linearity. Model singularities can occur if any of the contacting structure can "float" or "slide" (ie. exhibit un-strained rigid body motion) prior to the contact being established (and then iteratively finding a unique F=kx solution).

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 (or incorrect modelling) 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, to avoid free floating. The inclusion of contact friction can remove some degrees of singularity from a contacting structure. However, 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. This latter requirement remains independent of the direction of any applied loads, just as for Section 1.

Any "soft springs" should be soft enough to make negligible difference to the results. However, noting the problems identified in Section 4 above, the springs should not be too many orders of magnitude softer than the surrounding structure. Note that it is possible to adjust contact properties to create the effect of contact interactions in advance of the contact actually occurring.

6. Other causes of singularity errors

Simple oversights can also cause an FEA singularity error. Some examples include:

- Creating and displaying a constraint set, but forgetting to select it in the analysis setup.
- Running multi-case analyses and incorrectly choosing (or not choosing) constraints in sub-cases.
- Not including a stiffness (Youngs Modulus) for a material - although most FEA solvers will trap this during generation of the element stiffness matrices.
- Not including a thickness for plate elements - although most solvers will trap this during generation of the element stiffness matrices.
- Incorrect load direction in contact analysis (eg. parts separating and floating).

These brief guidelines have been created for Nastran users. However the same principles apply to other FEA codes, such as Abaqus, Ansys, Patran / MSC.FEA, Marc, Cosmos, Algor, Strand - and many more.

*** Footnote on the NASTRAN Grid Point Singularity Table. Singularity nodes/DoF, listed above UFM 9137, should never be confused with the (typically long) Grid Point Singularity Table. In plain English, the Grid Point Singularity table lists all the nodal DoFs which have zero stiffness contributed from the attached element(s), and thus completely eliminated from the matrix calculations - because zeros do not need to be solved as unknowns. For example, nodes of solid elements have no rotational stiffness, so RX, RY RZ rotation DoFs at all these nodes are eliminated from the {F}=[k]{x} calculations - EXCEPT at nodes where another element (if it includes rotation stiffness such as a plate or beam), is connected to the solid element node(s). DoFs at such nodes are no longer eliminated from calculations - but may well cause analysis singularity (lack of unique solution to F=kx) due to reasons detailed earlier - eg. noting that a beam element attached directly to a solid element node is functionally a ball joint.