Sub -modelling within an FEA Project

This Tech Tip looks at Global / Local Modelling (sub-modelling).

Sub-modelling principles can be used with FEA codes such as Abaqus, Ansys, Marc, Strand 7 and more. However the focus of this document is to demonstrate the procedure when using Femap with NX Nastran which provides a convenient, ergonomic technique for performing submodelling.

The typical reasons why sub-modelling may be of value in an FEA project include:

• A large assembly which includes small, but important, local details may need more mesh than is convenient or possible to solve with available computing power;

• Design changes for satisfactory or optimal behaviour might only be required in one or a few local areas of a much larger assembly;

• A global structure that behaves in a mostly linear fashion may have critical non-linear behaviours (eg. yielding) in local zones, which may need careful evaluation. In this case, submodelling may allow non-linear behaviour to be analysed in the local model, without subjecting the entire model to the rigours of the advanced physics, or without needing to use "super-elements".

Global-local analysis (sub-modelling) is valid and useful based on the following characteristics:

• Any individual element ultimately produces results which are a function of the displacements at its nodes, plus any elemental loads (eg. pressure, body loads). This useful fact extends to any subsection of a larger model. If we take displacement results at the boundary nodes of the subsection, and impose those displacements (and rotations) at the same boundary nodes and then throw away the rest of the global model, then the subsection (sub-model) is unaware the global model has gone. The results in the subsection will be identical to what they were when the global model was present.

• If changes are made within the sub-section, then the validity of sub-modelling depends on the condition that the stiffness of the sub-model observed at the boundaries changes negligibly for any changes made within the sub-model. For example, if a sub-model consists primarily of 3mm aluminium plate elements, then a sub-model design-change could not cadimageplm.com 3 validly change most of the elements to 5mm plate and/or to steel, as the stiffness observed at the boundaries is substantially changed. However, if addressing stress concentration around (eg) a 50-150mm hole in the middle of a 1000mm sub-model, then quite a large range of configurations could be used which would not significantly change the stiffness observed at the sub-model boundaries.

• Similarly, model non-linearities, such as localised yielding could be investigated from a linear global model if the yield effects (or any other non-linearities, such as large displacement or contact) are localised or mild enough that the force-stiffness behaviour at the sub-model boundaries are changed negligibly.

It is clear that if sub-modelling can be used, then it has the benefit of saving potentially hours or days of runtime. If a global model is 1m nodes, and a detailed sub-model can be extracted with only 50,000 nodes, then runtimes and design change investigations could easily be accelerated by factors of >100x. The productivity benefit is particularly large if the physics involves yield or contact, both of which might change a complex global model into being inconvenient or unworkable.

Sub-Modelling in Femap

So what is the key procedure in Femap? This document covers the simplest method, where the boundary nodes of the sub-model remain unchanged from the global model. It is possible to do sub-modelling where the complete mesh (including boundary nodes) can change between the global model and the local model, but this document covers the simplest case.

For each global result case for which sub-model results are required:

(a) Create a new load set which corresponds to the original load set that produced the current results. It may be easiest to simply make a copy of the original load set (right click in the Model | Info Tree), and delete any load definitions that are not relevant to the sub-model. This depends whether load combinations or sub-case combinations have been used. Making a copy (or using the original set) has the advantage that body loads or other load definitions which relate to the sub-model, do not need to be re-created (and can be edited if, for example, a multi-surface pressure is now only applied to a few surfaces in the sub-model).

(b) Model | Load | From Output -> Displacements. Un-tick the "Select All" checkbox shown right before pressing OK, otherwise the loads will be applied to every node in the model.
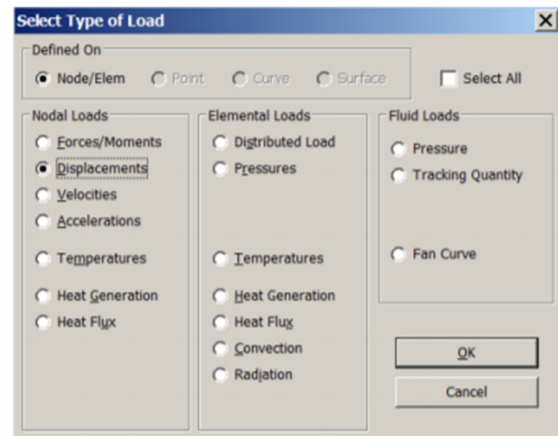
(c) The following dialog box appears, and it is simply a "join-the-dots" activity of connecting the translation / rotation results with the matching enforced displacement / rotation loads.
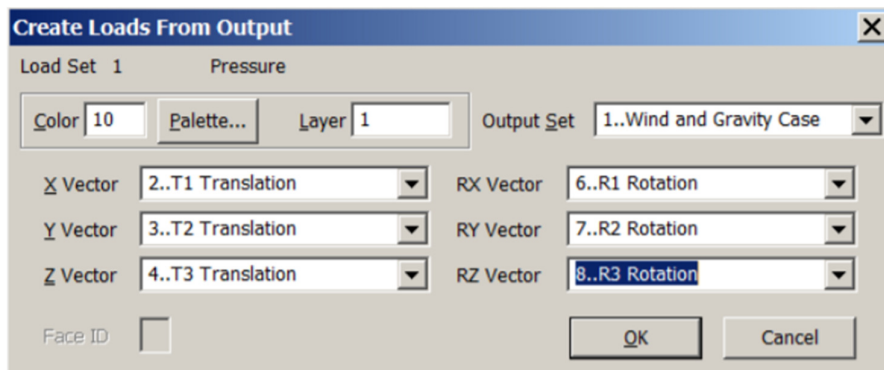
Then Femap asks for the nodes to which the enforced displacement loads are applied, which are obviously all the nodes at the external boundary(ies) where the sub-model is connected to the rest of the global model.

Once the enforced displacements/rotations have been applied to the boundary nodes, it is then possible to delete the remainder of the model, leaving only the sub-model. Keen users could arrange the model such that only the sub-model group is analysed, rather than delete the full model, but those instructions are well beyond the scope of this tech-tip.

There are two additional points to ensure perfect sub-modelling, relating to constraints and analysis precision (particularly if the structure is "stiff", such as steel, aluminium or concrete)1 . Note that when best practice is used such that the sub-model boundary is sufficiently removed from any local mesh refinement or design changes, it is usually easy enough to preserve the boundary nodes and include a quality mesh transition. If it is not easy to preserve the boundary nodes, this suggests the selected boundary location may be too close to the specific local area under investigation - and in particular if there are design-changes which cause minor local changes in stiffness. Also note that it is possible to use "force-based" sub-modelling in Femap with NX Nastran, rather than "displacement based" sub-modelling, however the displacement based approach is the easiest to apply.

Global model of a generic grinding mill

A model like this would typically include 13-19 rotating load cases, hundreds of pre-loaded bolts, contact and friction. The model size for this type of model is typically 1-2m nodes. Runtime for the full model of this detail is 40-80 hours on good computing.
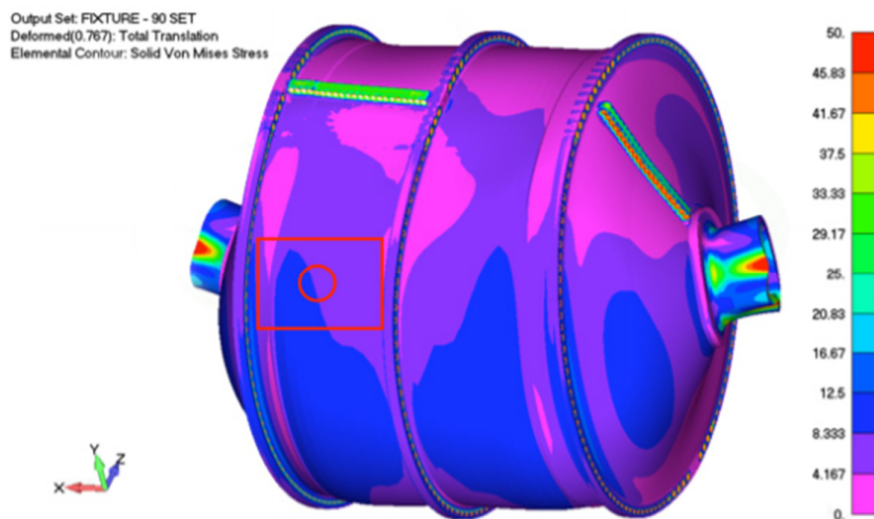
A "ball port" (=hole) is sometimes required in the cylindrical barrel of the mill. This local design detail can be considered without re-running the entire mill model, by using sub-modelling.
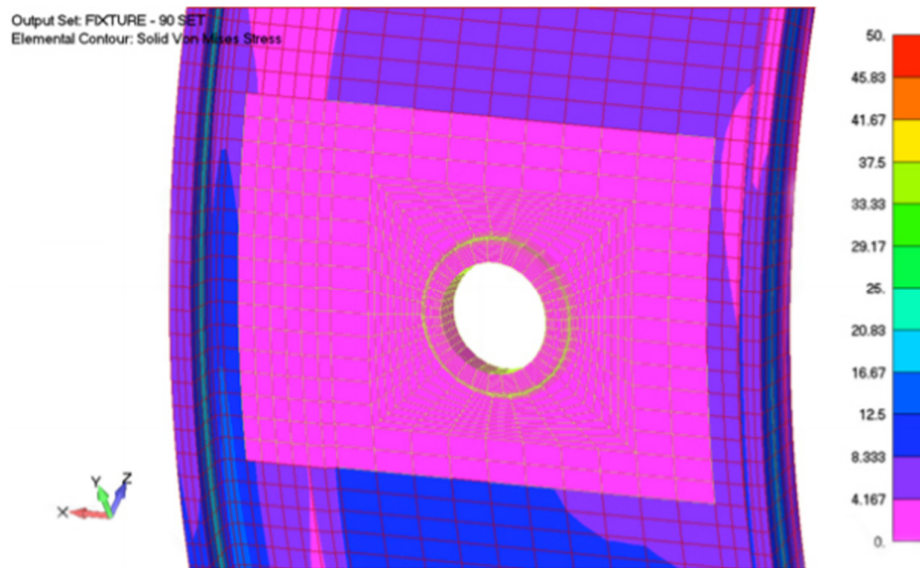
Above: Continuous mesh in the cylindrical barrel, replaced by local mesh including hole and local reinforcement (hence zero stress in the local mesh).

The global mesh is not deleted until enforced displacement loads have been applied to the boundary nodes of the local model.
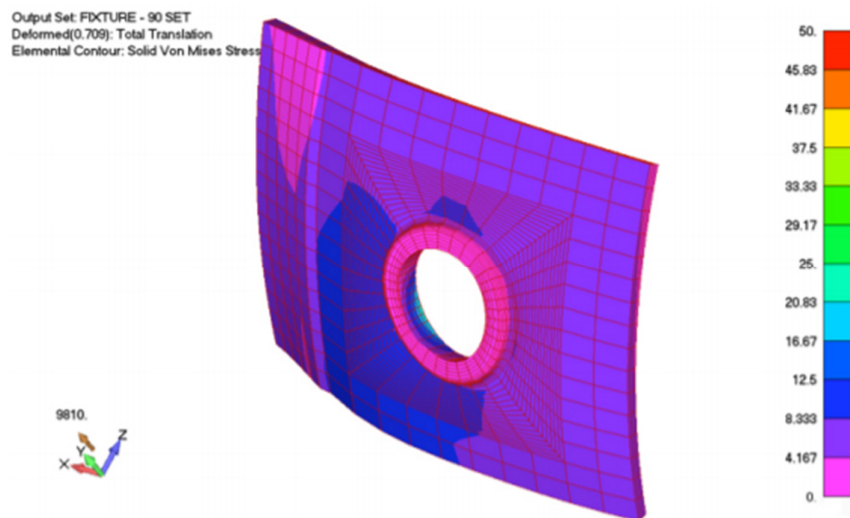
Results from local model of ball port (corresponding to load/result set shown in previous image).

Runtime for the local model (5,000 nodes) including 13 load cases 4,000 times compared to investigating changes in this design detail compared to the global model.