DURABILITY ADD-ONS FOR ANSA AND µETA

Dr. Dietmar Fels
Ford Werke GmbH / Germany

KEYWORDS – Durability, Scripting, Pre –and Postprocessing

ABSTRACT - The functionality of ANSA and µETA has reached an outstanding level. For beginners and also for experienced users it might be difficult to keep the overview about all options for pre- and postprocessing of finite element models. However user-friendliness and efficiency can be improved by the use of customized options to generate and check models and to accelerate the postprocessing of results.

The scripting language for ANSA and µETA is a powerful tool to add user routines. Especially the huge libraries of predefined functions allow the generation of source code in an acceptable amount of time.

A collection of tools for durability and strength has been created. A first group of scripts covers the check (including fixes) of models according to company internal guidelines. The pre- and postprocessing of spot welds has been improved by several functions (for example postrealization function for thickness dependent failure criterion; calculation of failure factors in µETA; section forces, stress components and fatigue of spot welds). The link between ANSA and µETA via named entities is a key feature for an accelerated postprocessing. ANSA tasks have been prepared to generate named entities for a standard postprocessing in µETA (e.g seat belt and bolt forces, buckle and submarining angles).

A special type of result mapping has been developed for sliding door slam simulation.

TECHNICAL PAPER -

1. MODEL CHECK IN ANSA

ANSA offers a wide range of options to check the content of an FE model. The usage of different FE-Solvers for specific analyses requires a certain sequence of checks.

The ANSA Task Manager is the right tool to combine standard and customized checks of a finite element model. In our durability department check tasks for NASTRAN, ABAQUS and LS-DYNA have been created (see Figure 1). The target of these check tasks is to avoid failures during the analysis run and to verify the compliance of modelling guidelines (element types, integration rules etc.). The tasks are focused on modelling issues that caused most of the failures during the analysis runs. The second item is the root cause for a customized solution.

The *Checklist* Functionality allows to sort the identified out-of-spec items. So fixing these issues can be focused on the worst items (e.g. for time step issues). For certain items special fix functions can be directly applied by a right mouse click.

Scanning of solver message files as NASTRAN *.f06, ABAQUS *.dat or LS-DYNA d3hsp/messag allows the identification of the failed nodes/elements directly in ANSA.

The scripts implemented in the Task Manager can also be used via the ANSA GUI as User Script Buttons.
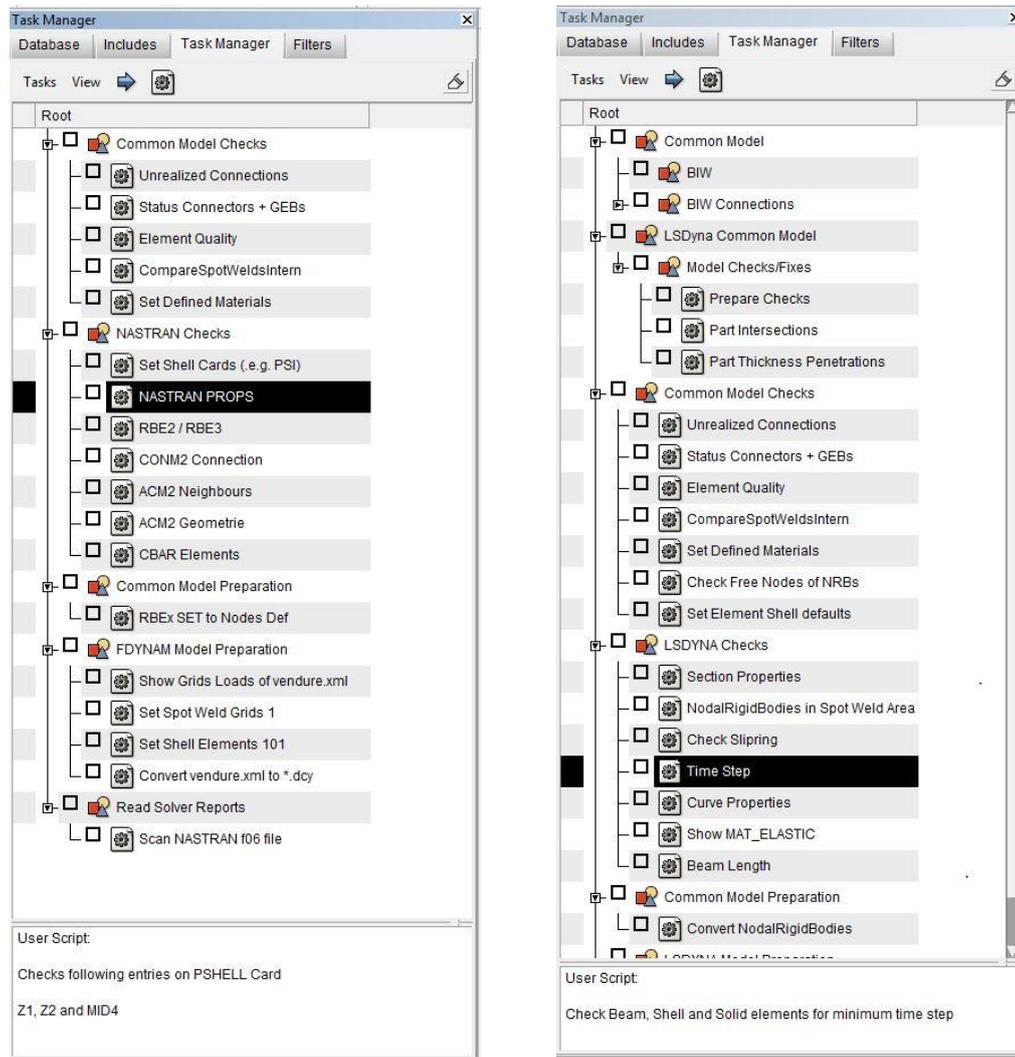
Figure 1 – Model Check with ANASA Task manager

2. LINK BETWEEN ANSA AND μETA

The evaluation of results for certain key elements and/or nodes is a common task. The identification of such key elements / nodes in μETA can be accelerated if these model entities get specific names in ANSA. This option was used in the tasks developed in the project presented in [5]. Especially for the ANSA entities connector and GEBxx with corresponding library items by one Apply operation the preparation for the postprocessing is completed (e.g. LS-DYNA DATBASE_HISTORY). The usage of names makes it independent of renumbering operations.

For μETA several scripts to report for example Bolt, Seatbelt or Joint forces have been developed. These scripts read the results, plot the curves and create annotations with the maximum result values. The usage of scripts saves a lot of time for this tedious process that may be required often during an optimization phase.

The in μETA detected hot spots (e.g. for damages, stresses) or regions where the results are probably influenced by singularities (RBE2s, ACM2 spot welds with RBE3 legs at feature lines) are exported as a list of hard points. As such this information can be directly read into ANSA to improve the mesh in the preprocessor.

3. SPOT WELDS

3.1 Preprocessing in ANSA by *Post Realization* function

This item has been started with simple modifications of RBE3 elements to be in line with fatigue tools – otherwise the spot welds could not be identified by these tools. The type RBE3-HEXA-RBE3 is the preferred modeling option for spot welds - mainly driven by its mesh independency. To check some potential modeling issues of this modeling option a few scripts have been created.

- If grids of the RBE3 elements are connected to feature lines or to triangular shell elements the results may be poor (e.g. false positive spot welds). This scenario is checked in ANSA and μETA. From μETA the information about such spot welds with high damages is transferred back to ANSA to control a remodeling of these spot welds. The check in μETA is done to filter the spot welds with high damage values. As such the fix in ANSA can be focused on these spot welds.

- If the dimensions of these spot welds do not match with the distances of the mid surfaces of the panels this may also give incorrect results.

One major objective was to allocate LSDYNA failure criteria to spot welds controlled by the thicknesses of the connected parts. The PostRealization function gets by default the following spot weld data from ANSA:

```
def BDuraPostFunction (element Connection, matrix XYZ_per_flange,
    matrix ProjEnts_per_flange,matrix interfaces_per_flange,
    matrix Bodies_per_flange_pair,matrix misc)
```

Based on this data all other needed information can be derived. The minimum panel gauge determines the allowable spot weld forces – with the exception that the thinnest panel is located in the middle. The *Post Realization* function is called for each selected spot weld. So for the 1st spot weld of the selection and/or ANSA session the spot weld materials are loaded from a material database. Also the corresponding contact has to be selected if more than one tied contact is defined in the model. This initialization is skipped later – controlled by ANSA variables.

3.2 Evaluation of LS-DYNA Spot Weld failure criterion in μETA

μETA offers the option to read all information of the LS-DYNA swforc file. So the time history of the spot weld forces can be plotted, and the failed spot welds can be filtered. To evaluate a model it's also important to know which spot welds are loaded to a borderline level. So the LSDYNA failure criterion is calculated for each spot weld by a script (the formula has been simplified by ignoring the terms with moments because the limit moments are not available, see [6]).

$$\left(\frac{\max(0,N_{rr})}{N_{rr_F}}\right)^2 + \left(\frac{N_{rs}}{N_{rs_F}}\right)^2 + \left(\frac{N_{rt}}{N_{rt_F}}\right)^2 < 1$$

The detection of the spot welds provides μETA as groups (if the LS-DYNA key file contains the ANSA comments regarding spot welds).The limit forces have to be read from the key file because the detailed material data are not available in μETA. As a result the time history of the failure factor of each spot weld is available (see Figure 2). The spot welds with a maximum failure factor **ff** higher than a certain level (e.g. **ff** > 0.8) are highlighted by an annotation in the 3D graphic window.
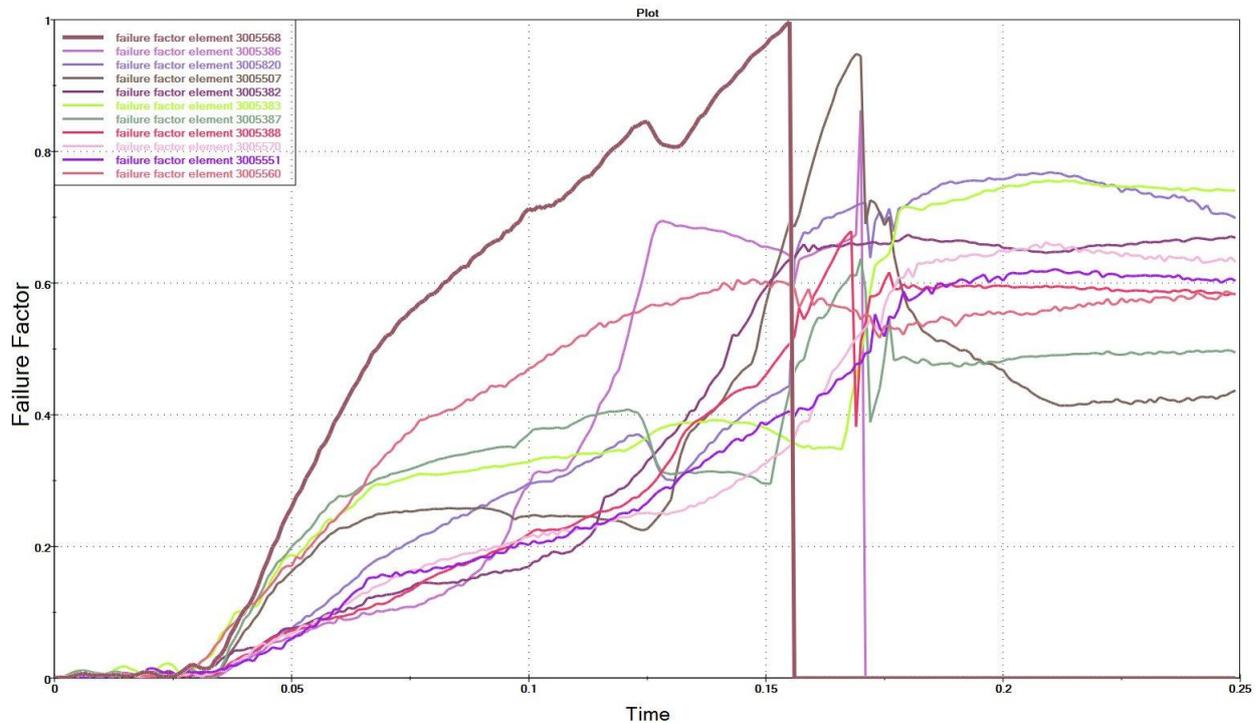
Figure 2 – Time history of spot weld failure factors

3.3 Section Forces and Stresses of Spot Welds

In contrast to the previous section this feature is not limited to LS-DYNA, it can be used for NASTRAN, ABAQUS and LS-DYNA.

For a better understanding of the failure mode of spot welds a group of scripts has been created to display section forces/moments and stresses of spot welds. Some interface work had to be done because up to version 6.6.x the *Grid Point Forces* from NASTRAN or ABAQUS could not be read directly into μETA (see Figure 3).

The script reads the Grid Point Forces, calculates section forces / moments and calculates radial stresses for selected spot welds. For each of these spot welds a separate page will be created that contains a zoomed region of the model, a 2D plot showing the section forces/moments and two polar plots showing the radial stresses (top and bottom panel).
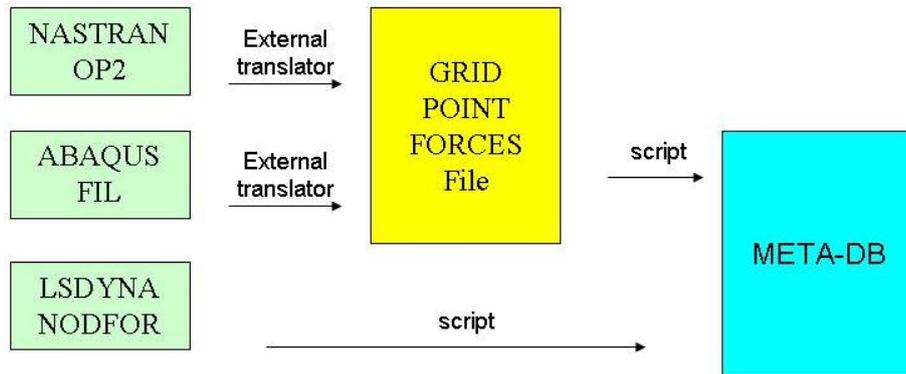
Figure 3 – data flow to calculate spot weld results in μETA

Figure 4 shows the results calculated for one spot weld. The orientation of the model (current view) is in line with the local coordinate system of the spot weld. So the axes of the polar plots are aligned with the model view. For selected each spot weld a new page will be created in μETA.
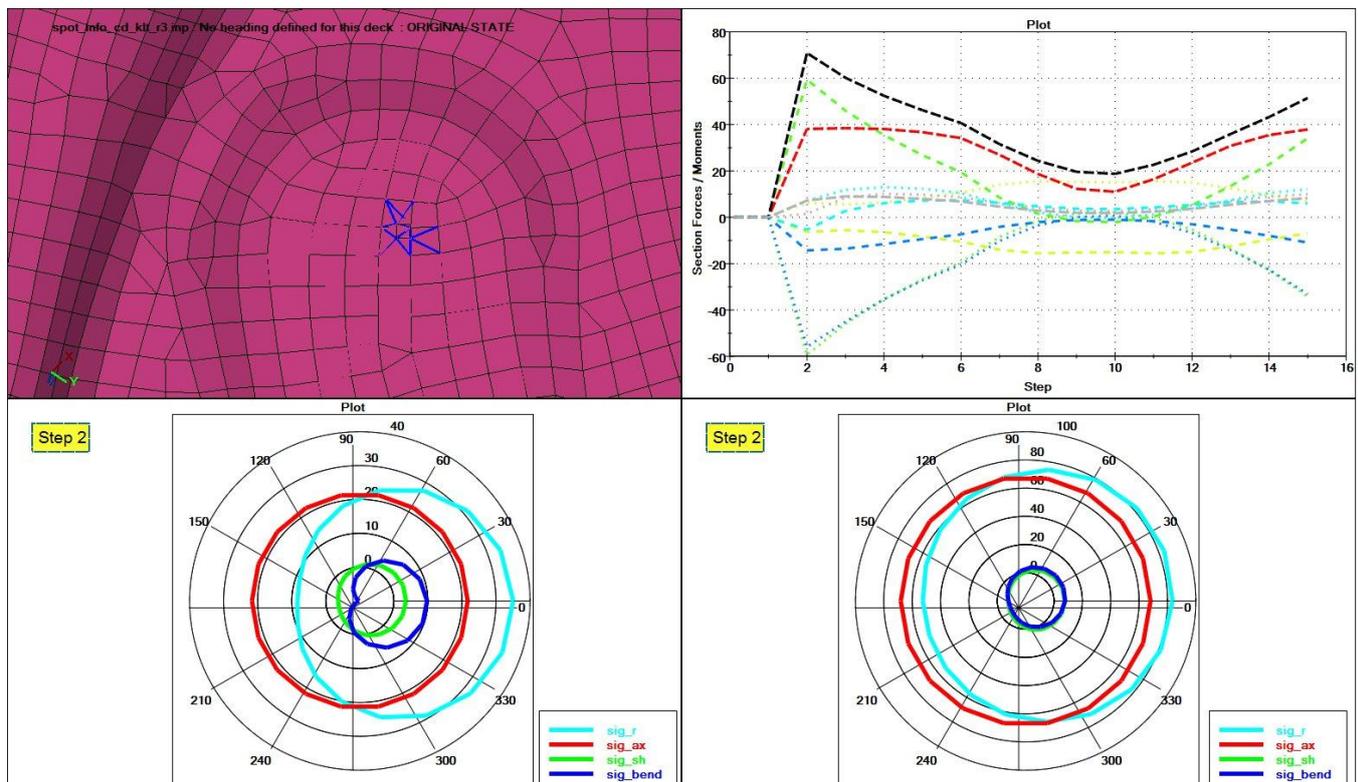
Figure 4 –spot weld results displayed in μETA

4. ANIMATIONS FOR FULL BODY FATIGUE

The frequently used quasi-static approach of full body fatigue analysis delivers damages for sheet metal and spot welds directly. To get a better understanding of the failure mode an animated FE model for time steps that cause high stresses is very helpful.

In general a quasi-static full body fatigue simplified consists of three steps

1. Linear elastic analysis with unit loads for different pay load conditions

2. Calculation of Stress Time Histories for different events (combination of pay load conditions and Road Profiles)

3. Fatigue Calculation for the entire test

To get a typical animation of the failure mode the following strategy is useful:

Find the Most damaging event for element / spot weld	→	Identity maximum stress with corresponding time step	→	Extract load vector and calculate displacements
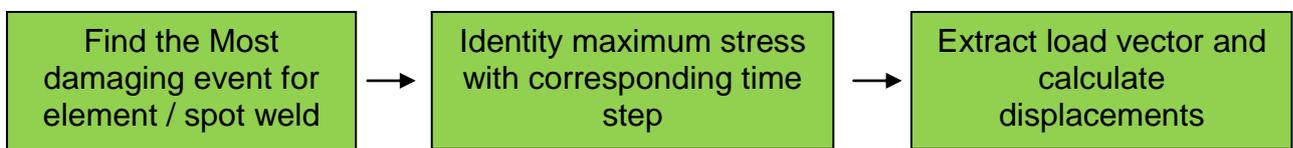
Figure 5 – Identification of critical load steps

The calculation of the displacements can be done by the solver (e.g. NASTRAN). To avoid the restart of the solver a μETA script has been developed that covers the above defined steps and the linear superposition. The displacements are exported in the linear static analysis for example into the NASTRAN OP2 file that contains also the element stresses and Grid point forces. The unit displacements, load time histories and grid point forces are read into μETA (see Figure 6). The linear superposition delivers the displacements for the interesting time steps.

For spot welds the selection of the time step is based on the maximum radial stress. For that reason the section force and stress time histories of the selected spot welds are recalculated in μETA.

A modified script allows the calculation of time histories of the distance and/or the difference of the displacements between certain points e.g. to determine the distortion of the liftgate or screen openings.
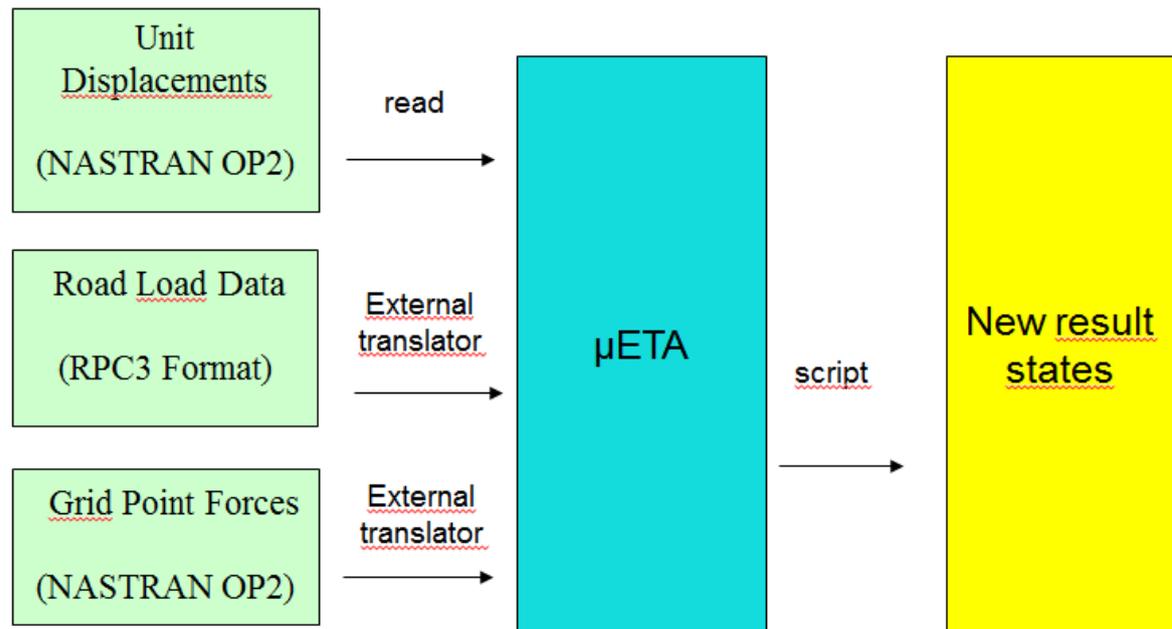
Figure 6 – data flow to calculate animations for critical time steps

5. SLIDING DOOR SLAM SIMULATION

The slam simulation of a sliding door is more complicated than for hinged closures. Compared to hinged doors, hoods, liftgates etc. there is no rotation around a fixed axis. So for a close slam operation a complete simulation from the open to the closed position has to be performed. With a default LSDYNA explicit analysis this may take a lot of time (about 40 hours). Using the *DEFORMABLE_TO_RIGID option this analysis can be significantly accelerated. So for the major part the simulation is done in the RIGID mode - just before the door is in contact to the door, bump stops and locking mechanisms. At this point there are two options:

- switch to the deformable mode in the current analysis

- stop the analysis and export coordinates and velocities, update the model with these data and start the analysis from this position with initial velocities

The RIGID mode of the first option may take also a few hours, dependent on the properties of the remaining flexible parts (e.g. springs or links). That's why the second option is of interest. Additional tools have been developed to support the second option. Coordinates and velocities are exported from μETA-DB (see Figure 7).

Figure 7 – export node coordinates and velocities from μETA

These files are read into ANSA for the current model status. The transformations for certain model entities (especially connections as spot welds, adhesive lines and bolts) are calculated and so these entities can be transformed to their new position.
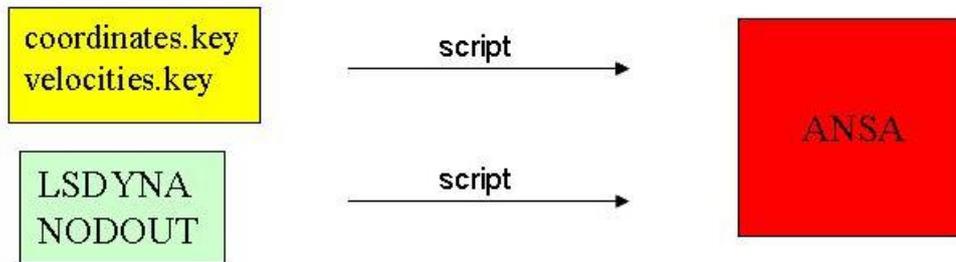
Figure 8 – model update for a certain time step

If the model will be modified (.e.g. new part geometry and mesh) for the new grids no initial velocities exist. Based on the information about the rigid bodies of the model for each point of a rigid body the initial velocities can be calculated.

$$\vec{v_Q} = \vec{v_P} + \vec{\omega} \times \vec{r_{PQ}}$$
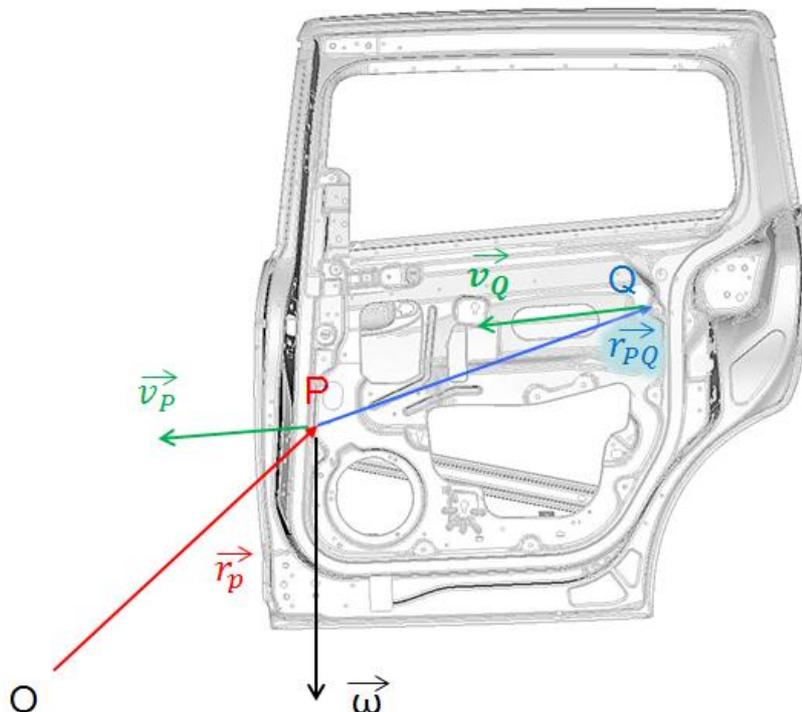
Figure 9 – calculation of initial velocities of any desired node dependent of a reference node of the rigid body

6. CONCLUSIONS

Starting initially with small scripts as aids for model check and result presentation a library of ANSA and μETA scripts has been created to support durability analyses has been created. The usage of the scripting language offers many options to customize ANSA and μETA. In the past , the function of the scripts was implemented using FORTRAN and C programs. Compared to these programs, a major progress is the direct visualization of the output in ANSA and μETA. The option to create a GUI for the scripts and the Checklist feature in ANSA increase the wide acceptance and easy take–up of new scripts.

The following proposed modifications would help to improve the scripts useability:

- An extended hardcoded data import in μETA (e.g. read Grid point forces, items of material data etc.)

- The announced option to have more than one result entity per result set in μETA 6.7.x offers more options (e.g. for fatigue calculations).

- To save variables and matrices (e.g. by script imported data) in ANSA and μETA would be a desirable option.

The closed Loop ANSA -> μETA -> ANSA has a big potential to improve the efficiency of the FE modelling.

The package has improved the efficiency of many working steps in the body durability department. Nevertheless in combination with the Task Package there is still a potential for further improvement.

REFERENCE

(1)   ANSA version 13.1.2 User's Guide, BETA CAE Systems S.A., February 2011

(2)   ANSA Scripting Language version 13.1.x, BETA CAE Systems S.A., February 2011

(3)   μETA PostProcessor version 6.5.0. User's Guide, BETA CAE Systems S.A., December 2009

(4)   BETA Scripting Language μETA PostProcessor version 6.5.0. User's Guide, BETA CAE Systems S.A., December 2009

(5)   LS-DYNA durability loadcases: Automated template driven process using the ANSA Task Manager, [1]Y. Kolokythas[*], [2]D. Fels[*], [3]M. Weinert, [1]BETA CAE Systems S.A., [2]Ford-Werke GmbH

(6)   Halquist, J.O., LS-DYNA Keywords User's manual. Version 971 R4, LivermoreSoftware Technology Corporation, Livermore, 2009.

(7)   DesignLife Theory Guide 5.1, HBM – nCode 2009**.**