

## STREAMLINED PROCESS FOR CREATING A RESULTS BASED OPTIMIZED MESH

**Emmanouil Kastrinakis**  
BETA CAE Systems S.A.

KEYWORDS –  
Mesh Refinement, ANSA, META

ABSTRACT –

This paper describes an automatic, although fully controllable process of optimizing and refining the mesh in structural models or assemblies for small deformation analyses. The proposed approach is based on the need to control how the areas to be refined are identified and remeshed. In order for this to be achieved, the refinement cycle should be extended from the barriers set by a solver and include a pre and post processor with advanced automation capabilities like ANSA and  $\mu$ ETA.

Setting up and driving the process through ANSA can result to the highest possible mesh quality but also to a better geometry description while minimizing the time needed for manual work. Additionally, areas containing boundary conditions or loads can be refined, in case these are parametrically defined. Including  $\mu$ ETA in the process lets the engineer to freely choose the method on which the identification of the areas to be refined is based by computing and estimating errors due to coarse meshing, while ignoring inevitable errors in areas where results cannot converge, between runs.

Finally, apart from the identification of the areas to be refined, the use of a post-processor with reporting capabilities, integrated in the process, can be used to inform the analyst either at the end of each cycle or at the end of the process not only concerning the refinement process but also regarding the development of a final complete report about the study.

The process is easily set-up in ANSA through a dedicated interface and it is open to any supported solver. Moreover, refinement of the mesh is accomplished by taking advantage of all meshing capabilities of ANSA while the identification of areas that need to be refined utilizes the filtering functionalities of  $\mu$ ETA which are based on multiple criteria and constraints. Thus, the refinement process becomes more robust and the final result is delivered easier and faster.

---

TECHNICAL PAPER -

## 1. INTRODUCTION

The procedure of refining the mesh of certain areas of a model, so as to yield stress results of acceptable accuracy, is common to CAE engineers involved with durability analysis and it is known in solver terminology as h-adaptivity [1]. This is an embedded functionality within some solver packages which however, is bound by the existence of certain constraints. In general this is a “black box” process with limited options for the specification of parameters being set beforehand. These parameters are taken into account by the solver which performs the refinement internally and delivers the final result.

Even if there are a lot of published studies in the literature focused in error estimation criteria [2] and mesh refinement algorithms [3], the current solutions lack of modern functionality required from analysts. These requirements are related to overcoming barriers set by a solver and it is already available in modern pre and post-processors. In the following pages of this paper, those requirements will be summed up and presented in detail. Then, an automatic, although fully controllable mesh refinement process implemented in ANSA and  $\mu$ ETA is presented along with an example showing that the use of a pre and post-processor can deliver a more robust refinement result easier and faster.

## 2. REQUIREMENTS AND CURRENT STATUS

The requirements of a mesh refinement process can be divided in two main categories. The first is related to the identification of those critical areas which need to be refined and the second is related to the remeshing of these areas. Additionally, there are some additional factors which could play an important role for an engineer, like the possibility to use the same process independently of the solver to be used and the creation of report displaying information about the refinement.

The following list briefly presents all requirements while they are presented in more detail in the following sections.

### Identification of critical areas

- Manual selection of areas
- Automatic selection of areas through single or multiple criteria
- User defined criteria calculated in the post processing phase
- Option to automatically exclude areas where results cannot be converged
- Option to inspect or modify selected areas

### Mesh refinement on critical areas

- Refinement should take in account CAD data and not be based on orphaned mesh.
- Use of advanced meshing algorithms which can produce state of the art meshing in complex models.
- Special treatment of geometrical features
- Option to inspect and modify if necessary the refined mesh
- Support of assemblies, reapplication of connector templates and boundary conditions as well as other entities such as sets for contact definitions

### General Requirements

- Solver independent process
- Automatic creation of report
- Meet all above requirements with the minimum effort and through an error-free process

It is clear that many of the above requirements cannot be met within a solver. A solver can do the refinement only in orphaned mesh which cannot lead to a reduction of the

---

discretization error in critical areas. Additionally, there are other limitations like not being able to control the quality of the produced mesh or not being able to define custom criteria (at least not in an easy and straight forward way) for the identification of areas to be refined. To overcome these limitations more complete CAE packages with extended functionality are required which provide to the engineer a more broad approach. In a full package is possible to control more parameters. Mainly there is the possibility to base the refinement on the CAD model description, automatically apply between iterations the boundary conditions and better control the criteria and areas to be refined [4],[5]. Yet, this is not adequate for very complicated models as the ones encountered in real studies. Such models impose high requirements in terms of meshing, both for the algorithms used and for the level those algorithms can be controlled automatically. Complexity is increased further and consequently the requirements from the refinement process become tougher if the model is assembled and thus, special treatment is needed for areas where connections and joints are applied.

### 3. IDENTIFICATION OF CRITICAL AREAS

Concerning the identification of critical areas, it is imperative that the analyst should be able not only manually to select the areas to be refined but to set and to combine a series of criteria for automatic identification. Additionally, the option to define custom criteria calculated by the post-processor and not the solver itself are often needed.

The commonly used error estimation calculations are based on various nodal calculations which vary from simply averaging from the neighbour elements to calculation discontinuities and errors [6],[7]:

$$\text{Average} = \bar{e}_i \text{ for } i = 1 \text{ to } N \quad (1)$$

$$\text{Maximum} = \max(e_i) \text{ for } i = 1 \text{ to } N \quad (2)$$

$$\text{Discontinuity} = |\max(e_i) - \min(e_j)| \text{ for } i = 1 \text{ to } N \quad (3)$$

$$\text{ProbableError} = \frac{\sqrt{\sum(e_i - g_{aver})^2}}{N} \text{ for } i = 1 \text{ to } N \quad (4)$$

$$\text{NormalizedMaxDifference} = \max\left\{\left|\frac{e_i - g_{aver}}{g_{aver}}\right|\right\} \text{ for } i = 1 \text{ to } N \quad (5)$$

$e_i, e_j$ : the requested for loading element results on corners or on integration points of the  $N$  elements connected to each node.

$g_{aver}$ : the average of the values of the requested for loading element variable of the  $N$  elements connected to each node.

Though, the identification cannot be based only on the above criteria. The engineer should be able to combine them with other information. For example, one would be interested in filtering areas where errors appear by taking in account also the stresses values.

Additionally sometimes there is the need to exclude from the identification of the areas, nodes which could provoke unnecessary repetition of the process. Specifically, concerning nodes on which constraints or loads are applied, it is possible that their values will remain higher than the critical value even after the refinement of these areas and in many cases they need to be disregarded by the engineer (Figure 1).

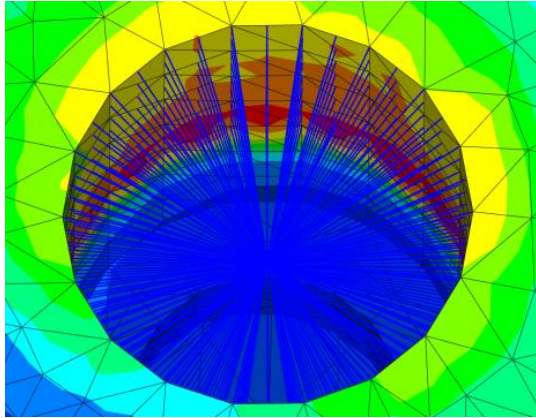


Figure 1 – Values on constrained nodes should not be taken in account when identifying critical areas.

Another requirement to avoid unnecessary further refinement is to omit nodes present in sharp edges or T-Junction from the identification of areas (Figure 2). Stresses in sharp edges tend to increase as the element length decreases [8].

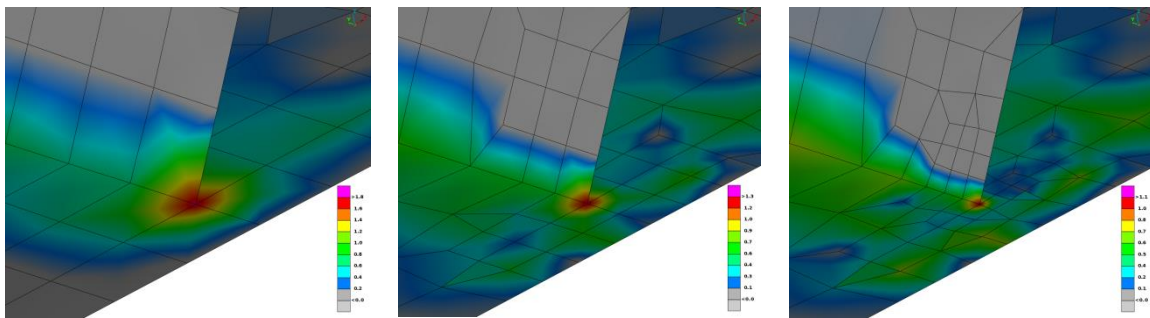


Figure 2 – Stress Singularity in T-Junction.

#### 4. DISCRETIZATION ERROR

Ideally, the presence of a model's geometry is preferred. In such a case the refined mesh will be based on the real geometrical features. On the other hand, refining an FE model (orphaned mesh) is limited to element splitting or remeshing but keeping frozen boundary nodes, while geometrical details lost due to coarse meshing cannot be reverted (Figure 3). The discretization error in this case cannot be reduced.

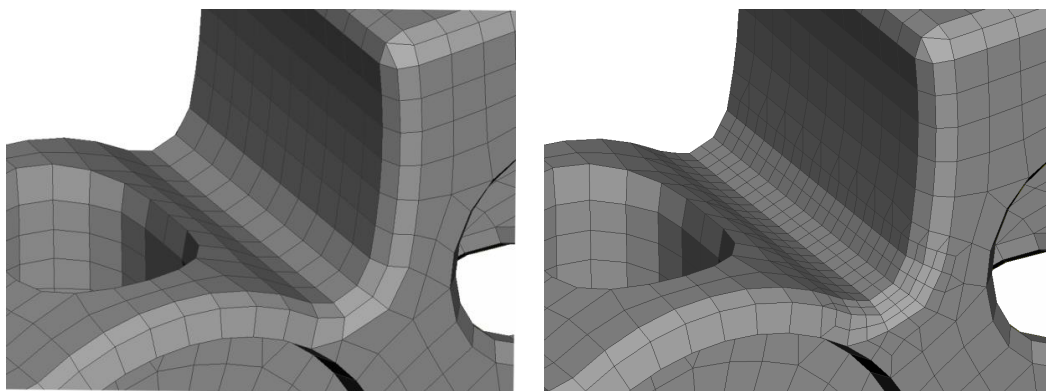


Figure 3 – Refinement on fillet based on orphaned mesh

---

On the other hand, having available the model's geometry, the latter can be taken in account during the refinement process and, depending on the element's size, some features can be reverted (Figure 4).

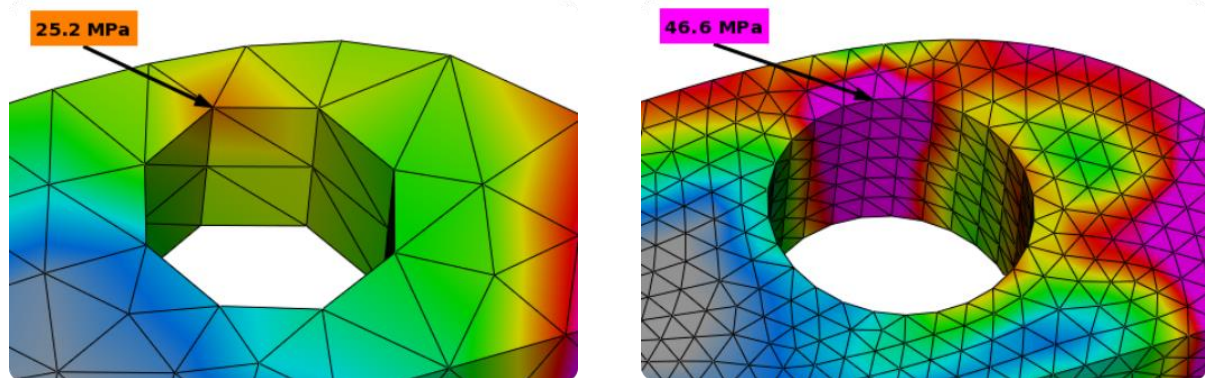


Figure 4 – Refinement on circular feature based on geometry

## 5. REMESHING ALGORITHMS

For a successful refinement, it is very important that the produced mesh falls within the engineer's specifications. It is possible that those specifications will vary from analysis to analysis, as they depend on several factors like the size of the smallest geometrical features to be kept or the modelling techniques.

Within a refinement process, the analyst has to be completely free to choose the required meshing algorithms, which can work on existing mesh reducing its size or which will recreate the mesh again with a smaller target element length. Additionally, often there is the need to apply special treatment on specific features like holes, tubes, fillets and flanges, for example creation of zones around holes, oriented trias in fillets etc.

Mesh refinement can be facilitated in ANSA through various options featuring a high level of control, thus producing sufficient flexibility. The simplest method to refine an area is through algorithms which recreate the mesh by taking in account the CAD model description and the previously element length of the area and reducing it by a predefined percentage at each iteration while at the same time respecting rules regarding the type of mesh, order, handling and treatment of geometrical feature like flanges, holes, tubes and element quality criteria thresholds which are used to improve the mesh quality.

On the other hand, in case that the current mesh needs to be taken in account, there are algorithms which work on existing mesh and they do not take in account a target element length but they produce elements of half the original size.



## 6. REAPPLICATION OF ANALYSIS SETUP

After the refinement of the areas, very often the areas to be refined include elements or nodes on which boundary conditions or other entities are applied. It is critical that the pre-processor does support the definition of entities like connections, sets, boundary conditions, loads or joints through templates which do not directly depend on the mesh. Reapplying them automatically on the new mesh saves valuable time and prevents errors. Otherwise, manual operations are needed.

This requirement is met in the proposed refinement process with the involvement of such entities in ANSA. These are entities whose definition is based on geometry (e.g. create a set of nodes defined by a face's edge) or through search patterns (e.g. create a rigid body connected to holes within a search distance) [9]. This information is stored in the ANSA database and can be used to automatically recreate them even if the mesh is modified (Figures 5 and 6).

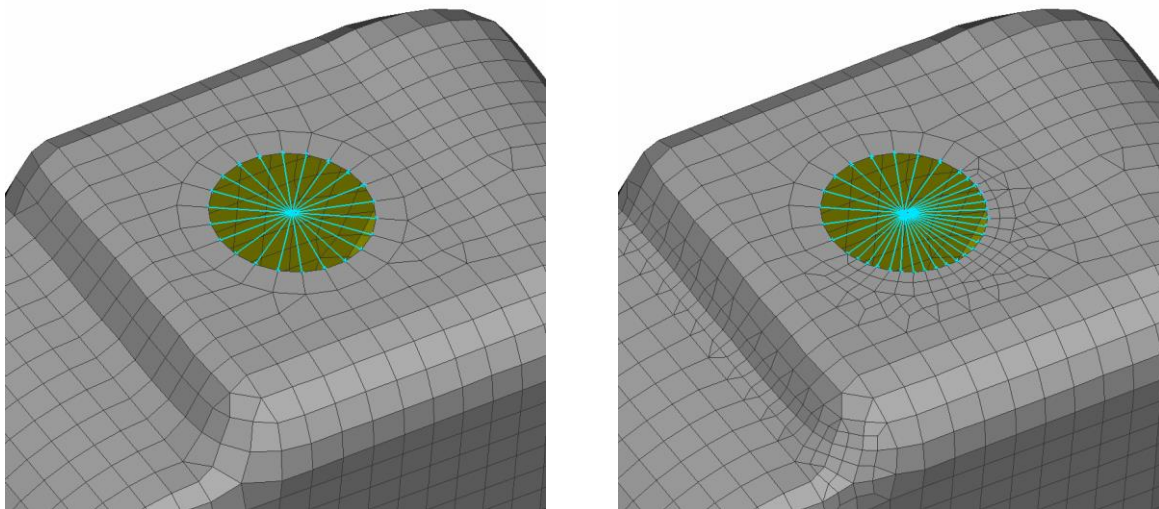


Figure 5 – Rigid elements recreated after refinement.

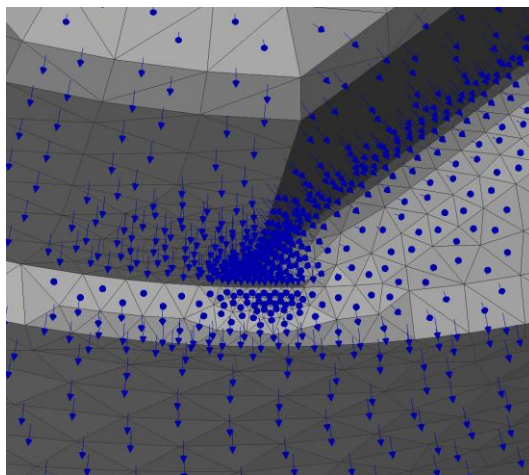


Figure 6 – Reapplication of pressure.

## 7. PROCESS OVERVIEW

To overcome all the impediments stated above, it is suggested that the whole refinement cycle (Figure 7) should be set up and driven by a pre-processor either fully automated or semi-automated, i.e. allowing the engineer to intervene if necessary.

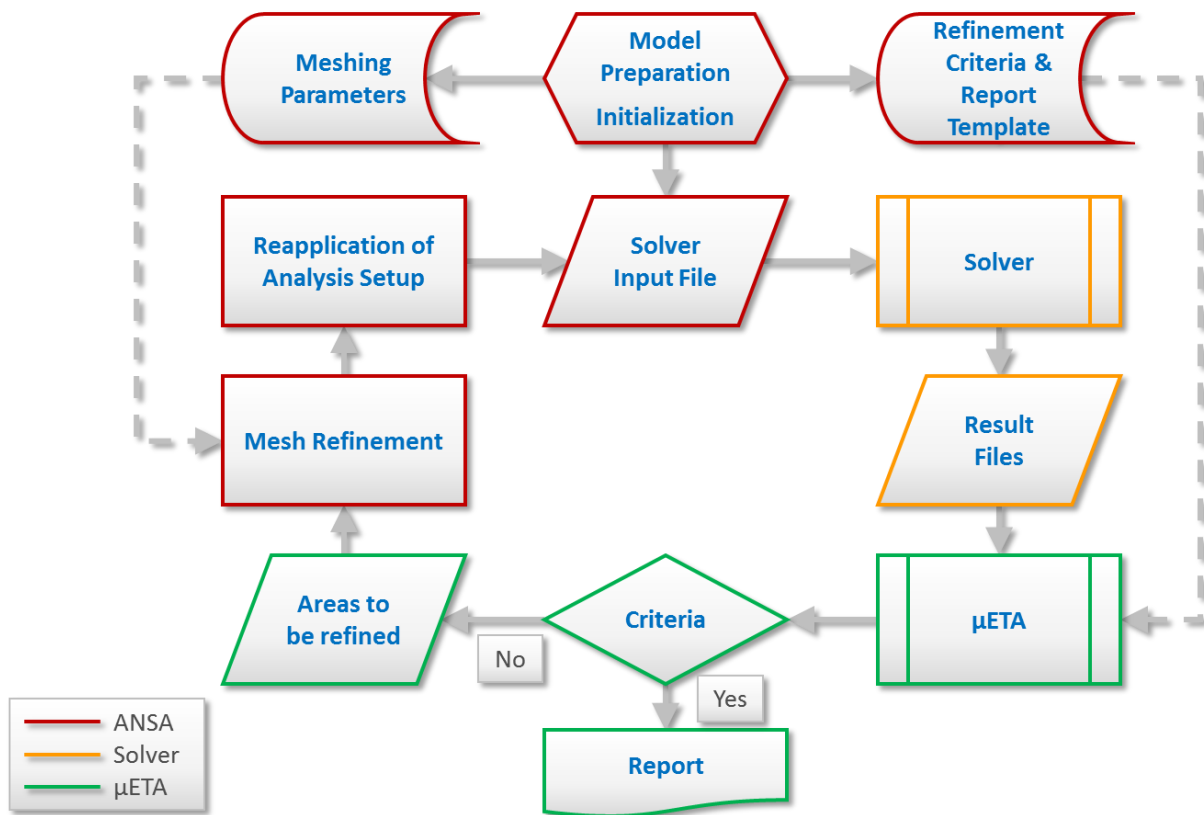


Figure 7 – Refinement procedure driven by a pre-processor.

A model can be input in a pre-processor as CAD data, native pre-processor database or FE model. The engineer is responsible for meshing the model, setting up the analysis and initiating the refinement process.

The prerequisites, in order to run the mesh refinement process, include all the parameters related to meshing, the criteria for the identification of critical areas, the maximum number of iterations and optionally a report template.

Once the refinement process starts, the pre-processor exports a solver input file and initializes the solver. The solution procedure is monitored by the pre-processor and if it finishes successfully, the post-processor is called. The result files are read and the areas which need to be refined are identified, according to the criteria previously set. If no such areas are found or if the maximum number of iterations is reached, a report is created and the process ends. Else, the identified areas are communicated back to the pre-processor.

At this stage the engineer should have the option to let the process continue without any interruption or pause, to review and to modify the areas to be refined.

Then, the preprocessor applies the mesh refinement. When finished, the analysis setup (application of loads, constraints etc.) is reapplied automatically, a new solver input file is exported and the cycle continues. Again, at this stage, the engineer should have the option to interrupt the iteration and inspect the result.

The process described above can be setup up through a simple purpose-built interface integrating all the necessary settings (Figure 8).

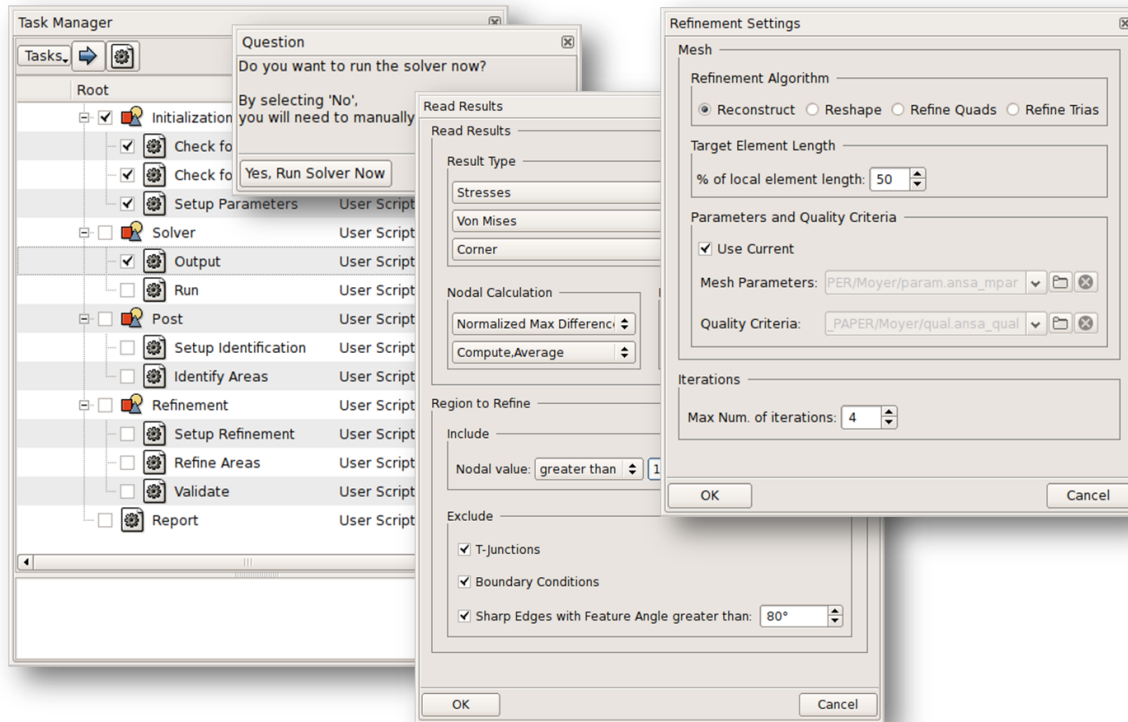


Figure 8 – The Auto Refinement tool in ANSA.

## 8. EXAMPLE: MESH REFINEMENT IN A BRACKET

In the following example, the refinement process is illustrated on a bracket whose CAD description is fairly complex (Figure 9). The main geometrical features of interest in this model are fillets and sharp corners. The purpose of the example is to demonstrate how to get accurate von mises stress results through refinement on critical areas using as criterion for the mesh convergence the Probable Error (see Section 3). The criterion is calculated in  $\mu$ ETA by reading stress results exported by NASTRAN.



Figure 9 - The model consists of a reasonable but still complex CAD geometry

A moment of 75kN is applied on the two upper holes while all other holes are fully constrained (Figure 10). All boundary conditions were created through a set builder, a special ANSA entity which can be automatically recreated after each refinement iteration. The initial



average mesh size was set to 10mm second order tetra elements, which is adequate but still not capable to accurately capture all geometrical features.

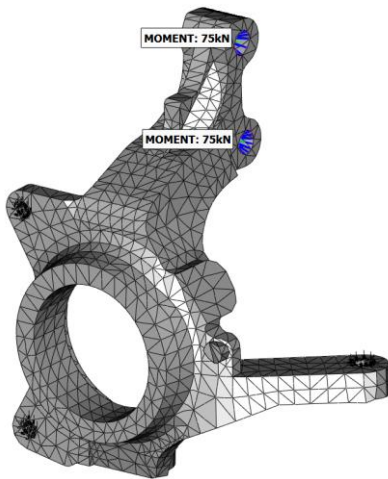


Figure 10 - Loading condition on initial mesh

The process was setup to run until the probable error dropped below unity in areas where the stress was higher than 20MPa, excluding areas on sharp corners after the second iteration. After the initial run, several areas with high stress values were recognized (Figure 11). In most of them, the probable error values were higher than the requested (Figure 12).

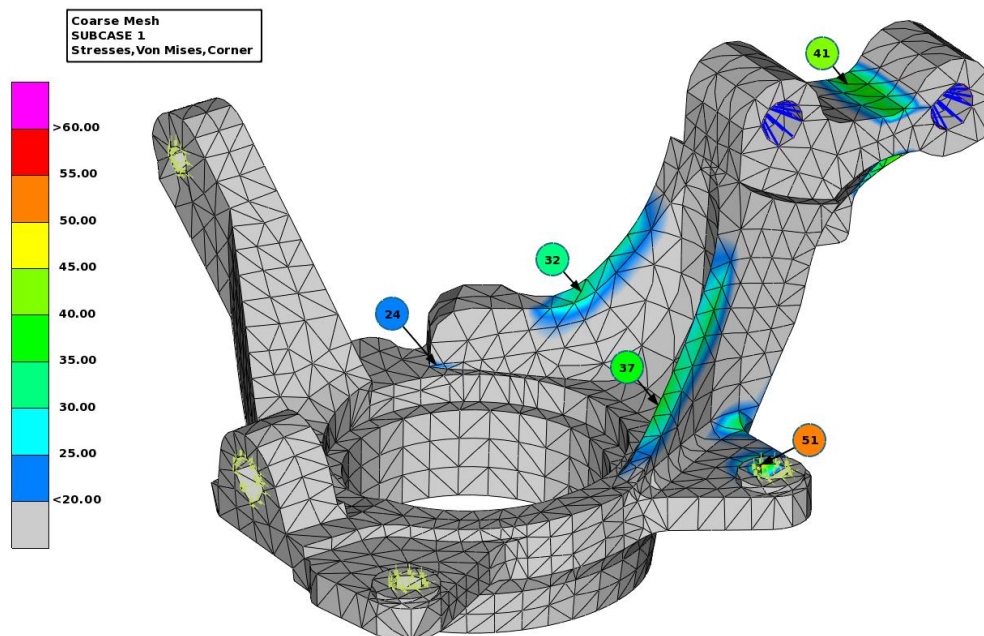


Figure 11 - Von Mises Stress results at first iteration.

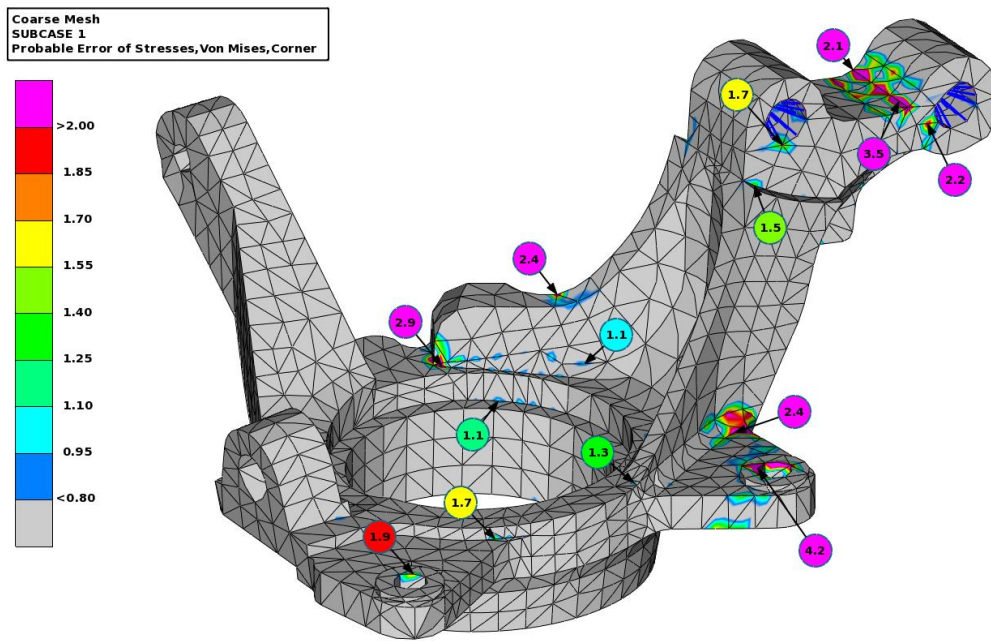


Figure 12 - Areas where probable error is more than 1 at first iteration.

The process is paused after solving the model for the first time and the areas to be refined are highlighted and the analyst is free to redefine them. At this point, refinement on sharp edges is allowed (Figure 12).

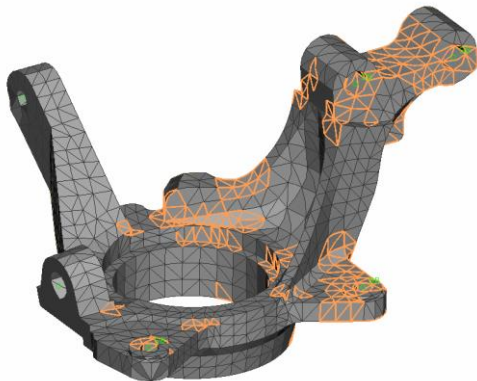


Figure 12 - If needed, the process can be paused to confirm or alter the areas to be refined.

Upon confirmation the areas are refined and a new version of the model is solved again. The probable error is reduced in some areas but further refinement is needed in some fillets (Figure 13). Additionally, at the next iteration, high errors which appear in sharp corners are ignored and no further refinement is applied on them.

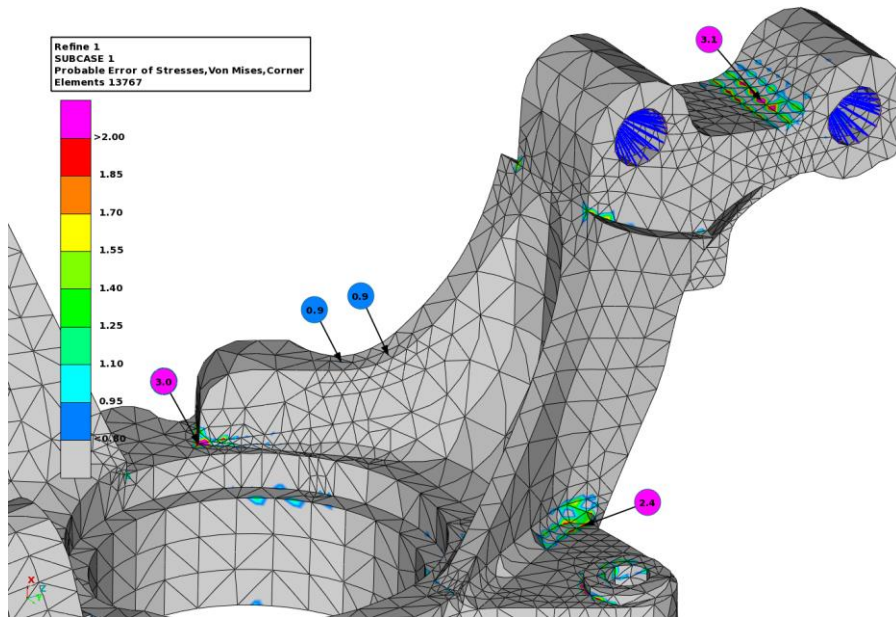


Figure 13 - Probable error after first refinement.

After four iterations the probable error is reduced close to zero (Figures 14, 15) at the areas of interest (Figure 16).

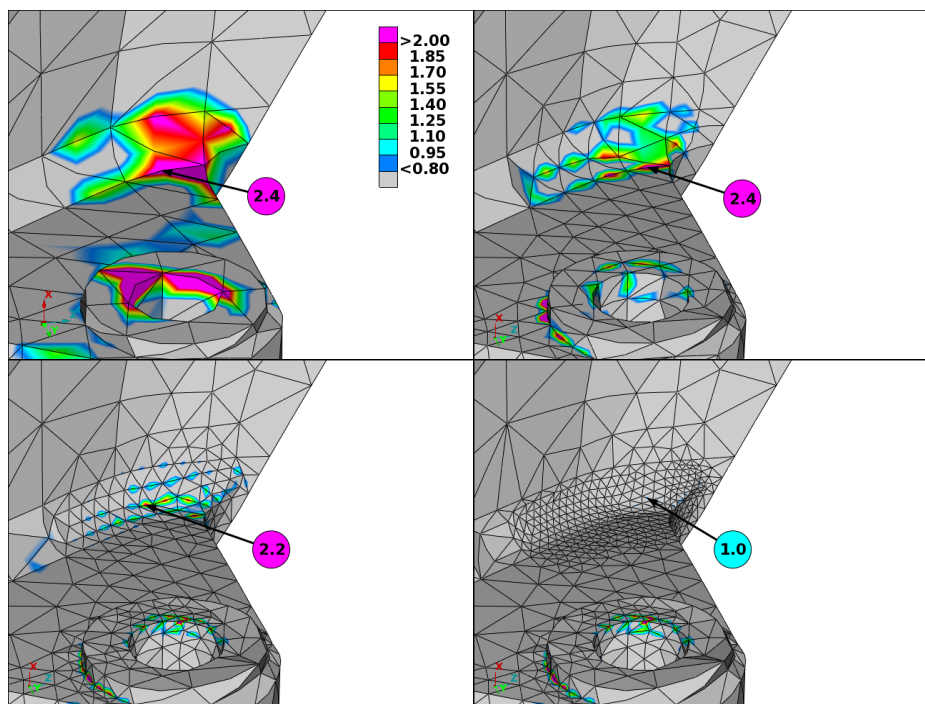


Figure 14 - Probable error values are minimized after four iterations.

Solution Number	Nodes	Elements	Probable Error (Von Mises Stress)
0	12294	6907	4.4
1	23419	13765	3.1
2	32049	19240	2.3
3	41919	25485	0.2

Figure 15 - Maximum Probable Error values, excluding sharp corners

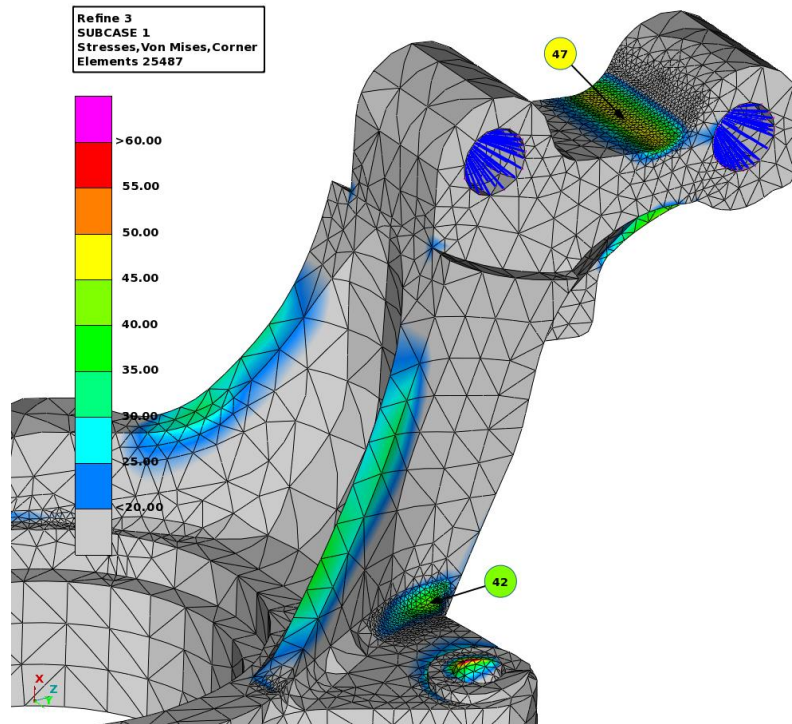


Figure 15 - Von Mises Stresses after final refinement.

At this point, it is worth mentioning that the element size at the refined areas is about 1mm and the total number of elements after all iterations reached about 25.000 tetras (Figure 16). Meshing the whole model with a 2mm element length is producing about 240.000 elements.

## 9. CONCLUSIONS

This paper presented the requirements for a successful automatic mesh refinement, not based in converging stress results between iterations, as this is not always possible in complex geometries, but through error estimation criteria combined with the option to exclude areas based on geometrical features. This can be achieved by using advanced tools found in modern pre and post-processors.

Realization of such refinement process within ANSA &  $\mu$ ETA package verified, that even for relatively small models, the analysis time to achieve accurate results is reduced. That being achieved with the least possible increase in the number of elements and with minimum time spent assuring, however, an error-free procedure. These benefits are magnified if the models size and complexity increases.

Finally, such process could also be implemented and driven by a Simulation Process Manager tool, like BETA CAE Systems' SPDRM. By taking advantage of its process evaluation features [10], it could be utilized as an optimizer for the meshing algorithm's parameters and quality criteria themselves. Some parameters could be set as variables and, in combination with the error due to mesh modeling, their optimum values can be found. Furthermore, additional automatic steps, not only related to meshing, could be included in the procedure.



## REFERENCES

- (1) Li LY, Bettess P (1997) Adaptive finite element methods: a review. *Appl Mech Rev* 50:581–591
  - (2) O.C Zenkiewicz, and J.Z. Zhu, “A Simple Error Estimator and Adaptive Procedure for Practical Engineering Analysis”, *International Journal for Numerical Methods in Engineering*, Vol. 24, pp.337-357, 1987
  - (3) Matthew L. Staten, “Selective Refinement of Two and Three-Dimensional Finite Element Meshes”, Masters Thesis, Brigham Young University, 1996
  - (4) Dassault Systems Simulia Corp (2011). “Abaqus CAE Users Manual”, Section 17.12, USA
  - (5) Dassault Systèmes SolidWorks Corporation (2010). “An Introduction to Stress Analysis Applications with SolidWorks Simulation, Instructor Guide”.
  - (6) BETA CAE Systems S.A.. (1997). *Element Results and Their Handling in META, META v.14.0.x User's Guide*. Thessaloniki. 813
  - (7) MSC Software, *MSC NASTRAN 2004 Reference Manual*, USA
  - (8) NAFEMS. The Importance of Mesh Convergence - 2. Available: (<http://www.nafems.org/resources/knowledgebase/002/>. Last accessed 20 Jan 2013./)
  - (9) BETA CAE Systems S.A.. (2013). 16.9. Generic Entities Builders (GEBs). *ANSA v.14.0.0 User's Guide*. Thessaloniki. 1100
  - (10) Stylianos Seitanis, 2009. Simulation Process, Data and Resources Management (SPDRM), 3rd ANSA &  $\mu$ ETA International Conference. Halkidiki, 9-11 September 2009, Greece
-