

Session H2.1

BUILDING DISCIPLINE SPECIFIC FE-MODELS WITH THE COMMON MODEL CONCEPT

George Nikolaidis*, Dimitrios Zafeiropoulos*, Konstantinos Ntamagkas
BETA CAE SYSTEMS S.A., Greece

KEYWORDS - FE Simulations, Discipline Models, Data Management, Batch Mesh, Representation Manager, CAE process workflow

ABSTRACT - The research and development in the automotive industry is faced with a continuously increasing number of FE simulations. Almost 75% of the total time required for the model building of a vehicle for 5 different disciplines, is spent until the engineer is able to apply all kinds of load cases to the analysis model. Therefore, the big challenge is to establish a pre processing workflow, which ensures that the different kind of requirements are met by these models, with a high reliability and reduced time and cost effort.

This paper presents the Common Model concept, introduced by BETA CAE Systems S.A towards the creation of Discipline Models starting from CAD data. A centralized Data Management system is proposed, to store, retrieve and provide engineers with all necessary data for the generation of the different models requested. The combined use of a Batch Mesh and Representation Managers will fully exploit the storing advantages of the proposed Data Management structure, keeping the overall CAE process cost efficient and error proof.

TECHNICAL PAPER -

1. BUILDING THE DISCIPLINE MODELS

1.1. Introduction to the CAE process workflow

The recent advances in parametric design and new CAD tools that have been introduced helped CAD departments not only to improve design work, but also to greatly reduce the amount of time required to produce large number of vehicle variations. Thus, it has been made equally important that the CAE engineering teams are equipped with the appropriate toolkit to deal with the increased number of model versions and load cases that should be analysed in an error proof, time and cost efficient way.

The basic concept on which the proposed way of dealing with the increased CAE work load is based, is that the creation of all Discipline Models that are to be analysed by the specified solver afterwards, depends on a pure geometric form of the model under consideration. The latter, also referred to as the Common Model, will have the correct Engineering data applied, such as Material and Property Thickness, as well as the welding information between parts. As soon as the CM has been freed of all possible design errors, it is ready to be meshed based on different scenarios in order for the requested Discipline Models to be produced.

All requests are passed through the Representation Manager, which will, in turn, coordinate the Batch Meshing process. The resulting model can be considered as the Discipline specific Common Model, which will serve as the starting point for all the appropriate load cases to be applied. The final analysis models are then sent to the specified solver for further analysis. As depicted in figure 1, the procedure proposed, is highly supervised by an integrated workflow manager, a tool where all the individual tasks of the development of a vehicle simulation model are included.

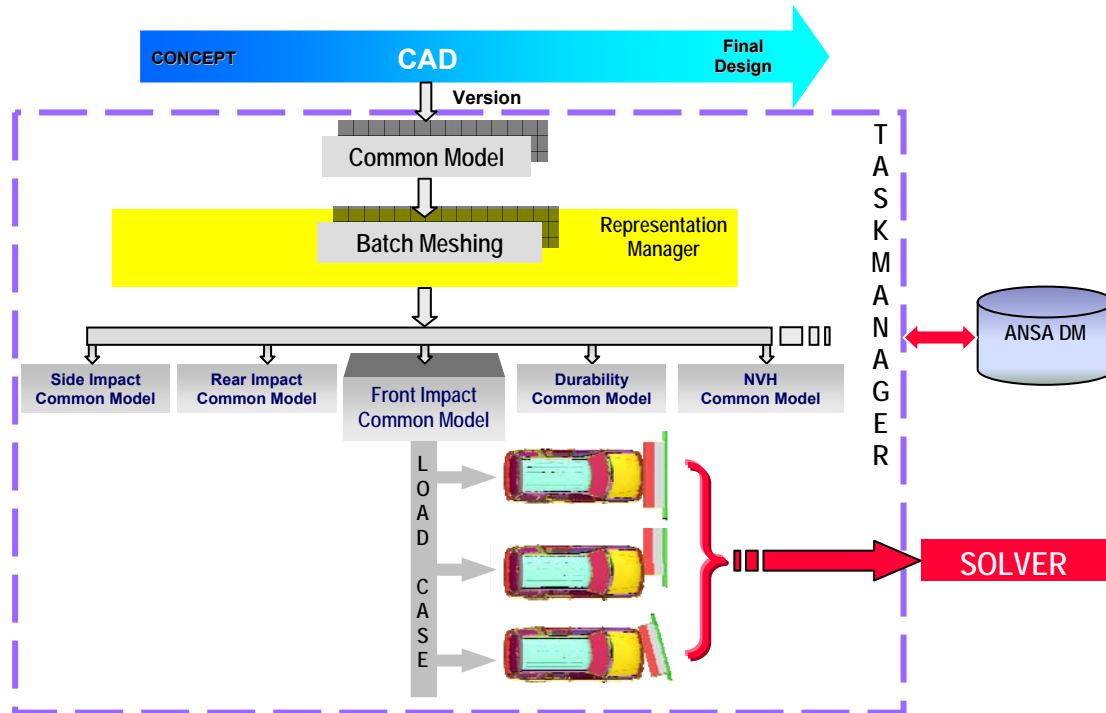


Figure 1

To facilitate and precipitate the CAE analysis, the Task Manager is in constant communication with a centralised data management system called ANSA DM, used to store, retrieve and monitor model components for potential CAD updates (1). The previously described procedure, assures that all CAD engineers work on the same and up-to-date components, the whole process workflow is error-proof and highly automated and that no time is wasted on performing the same tasks twice. The complete process as well as ANSA DM and the different parts of the Task Manager are described in more detail in the paragraphs to follow.

1.2. ANSA Data Management (ANSA DM)

ANSA DM is a structured and centralized Data Management System. It assures that all engineering data, which are used during the building process of a vehicle simulation model, are securely stored and can be retrieved at any time by all CAE engineers, working on various fields of development, (2). Typical data residing in DM are:

- The individual sub-systems, such as seats/dummies, walls/barriers, in their FE representations.
- Library items, such as connectors, boundary conditions and output requests.
- Files containing engineering, connection and trim data.
- Load cases, such as Front, Side and Pedestrian impacts.
- Material data and
- A number of different custom items.

In the case of Part and the Subassembly Data, information is organised in a structured way, corresponding to the output format of the CAD departments. Different CAD versions are stored in such a way, so as to follow the release sequence of the designing departments, (3). At the beginning, each version contains the part's pure geometric description, as produced in the design phase of the vehicle development, also known as the part's Common Representation (CM). The latter will be the basis for the creation of all Discipline dependant mesh variations, which in turn will be used for the production of all requested Reduced FE Representations, such as Lumped Mass or Trim.

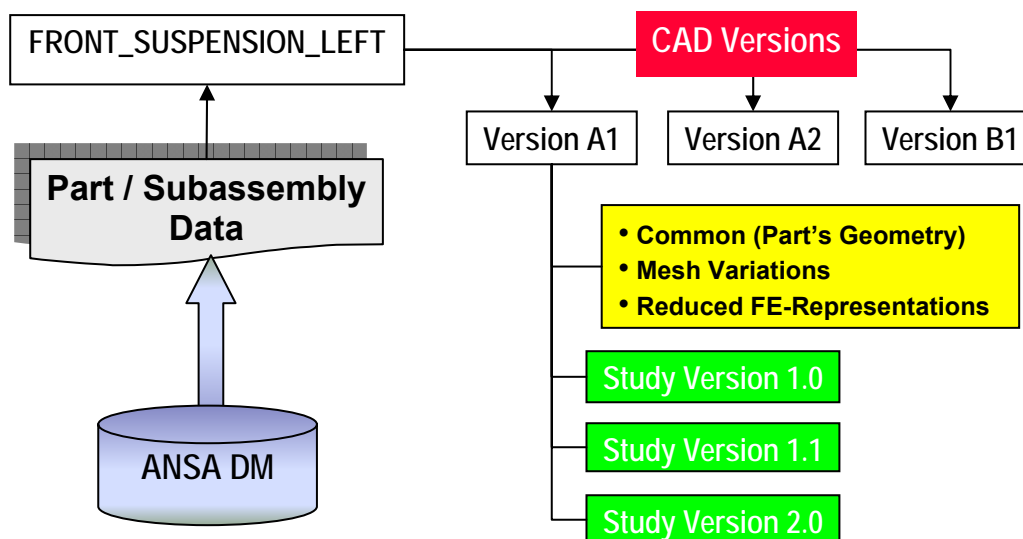


Figure 2

The first FE representation files to be produced in this way can be considered to form the First Study Version of each part. The Optimization Process uses the first Study Version as starting point. All other representation models that are to be developed constitute optimized variations of the first study version. They will thus be stored accordingly as later study versions of the same part, as depicted in figure 2 above. Once saved in ANSA DM all part versions and study versions are available to engineering teams for reference and further optimisation. Through DM, CAD version changes are effectively monitored, while CAE teams are notified the moment component updates are available, thus greatly facilitating all optimization work.

1.3. Building the Common Model

The designing output of the CAD department constitutes the model's pure geometric form. In order for the CAE engineers to begin working on it, this form must first undergo a certain translation process, which will lead to the production of the Common Representation, as previously mentioned.

The first action to be performed is the Definition of the Model as described in the VPM tree, a file exported by the CAD system and handled over to the CAE department. The VPM Tree Editor, (figure 3), will not only distinguish all different parts but also recognise the Symmetry and the Multi-Instantiated ones, as parts being at different locations. Translated parts that have already been saved in ANSA DM are also recognized. As a result, from the group of parts forming the model, only the newly added ones are exported to a Download List (D/L-list) to be translated.

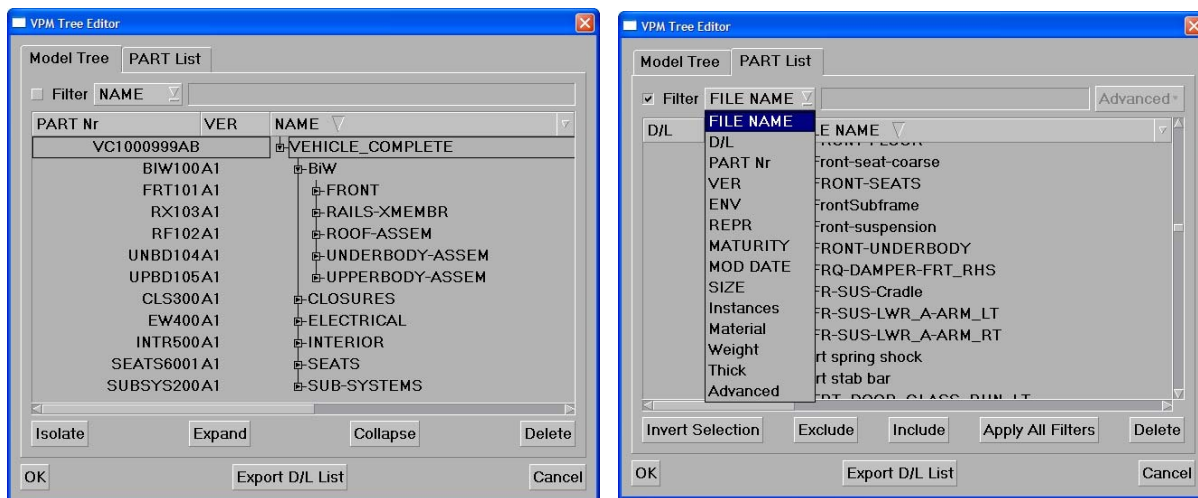


Figure 3

The next Step to be taken is the translation of the CAD data, for which the D/L – list works as an input. A significant number of issues is taken under consideration, to ensure that all parts are correctly translated. To begin with, the part offset can be performed directly in the case of sheet metal parts in a thin description, if the Part's Offset Rules are included in the CAD file. To continue, the middle-surface will be extracted in the case of sheet metal parts in a solid description. As far as solid parts are concerned, a volume can be defined automatically. Also, correct Engineering Data, retrieved from the D/L - list can be applied during the Translation. It is also worth mentioning that any arising errors and any missing-information will be noted as comments and will accompany the translated part in the outputted file. Comments can be later reviewed, and help with the detection and correction of errors.

After translation has been completed, the imported parts are positioned as described in the VPM tree to form the complete model. Parts that had been previously translated are retrieved from DM, in their latest CAD version and are positioned along with the new ones. All Welding Information will be imported at the end, to conclude the model's assembly process. The last step of building the CM consists of checking the model in order to fix any translation or assembly errors. ANSA automatically detects all wrongly positioned parts and the in-accurate welding information and offers all the proper tools to deal with them.

As soon as any errors are resolved, the Common Representation of the new parts is stored in ANSA DM and can be used further on for the creation of all FE representations. At this point it should be made clear that if possible errors are not fixed at this moment, engineers will come across them in each Discipline Model. The error free Common Model, once built, can be used as a starting point for the creation of each Discipline Specific Model.

1.4. Representation Manager

Once the Common Model has been built, all needed Representations can be constructed through the Representation Manager (RM), which handles all Representation requests for each Part or Subassembly, (3).

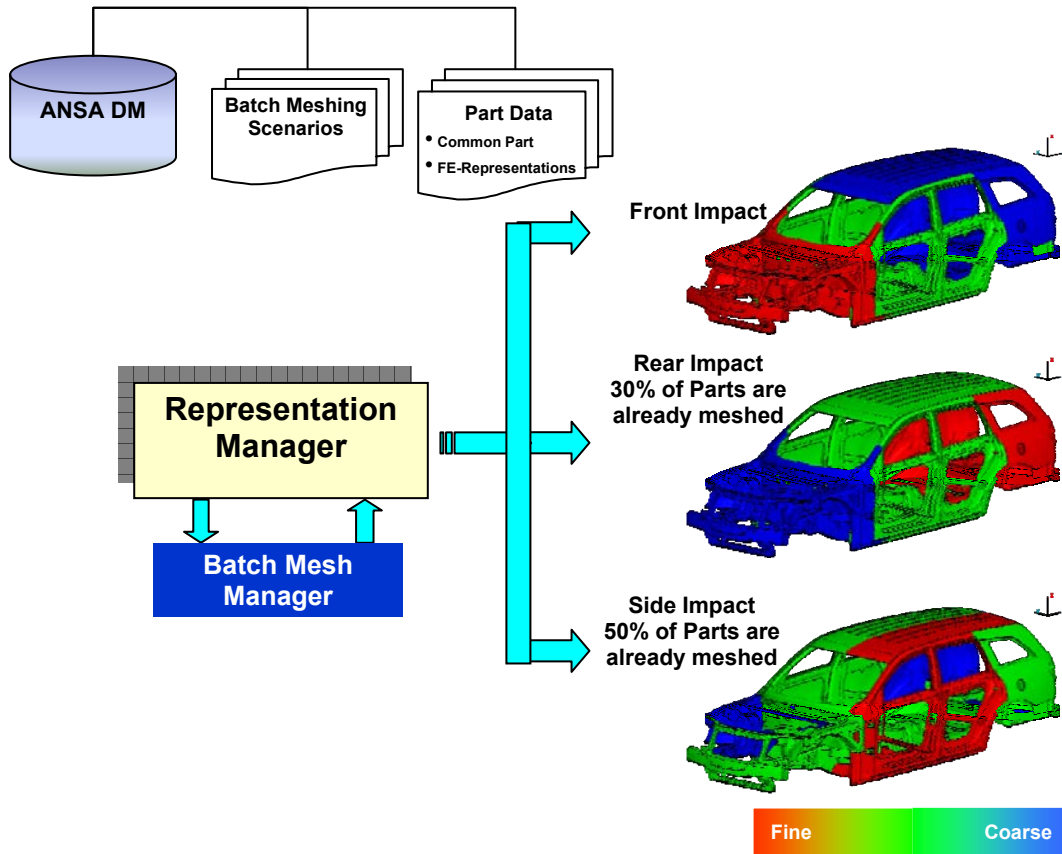


Figure 4

The moment a specific Discipline FE Representation will be requested, a search is carried out in DM. In case no such Representation has been saved, RM will retrieve the common representation and the respective meshing scenario and will send the job to the Batch Mesh Manager (BMM), which is designed to execute the different Discipline Meshing Scenarios. Each BM Scenario consists of a set of Meshing Rules. Parts according to their characteristics, such as location and name, will be meshed using the predefined Mesh Parameters. This concept can be applied either on the whole model or on single parts. RM will commit each FE-Representation in ANSA DM, and at the same time will record the specific Discipline this mesh was created for and the meshing rules that were used. This procedure assures that no meshing scenario is executed twice

In the example illustrated in figure 4, the Representation Manager will return upon request, the Front Impact Model. One should notice that the mesh size in the front region of the model is smaller than in the rear one. In the case where the Rear Impact model is requested, meshing will not be performed in all parts. Since 30% of parts have been already meshed with the same scenario for the Front Impact representation, they will be directly retrieved from ANSA DM. Only the remaining parts will be sent to the BMM to be meshed. The same principle applies in the case of the Side Impact Model, where due to the common meshing rules applied, 50% of the parts have already been meshed in the execution of different representation scenarios.

1.5. Dealing with new CAD versions

As soon as a specific Representation is stored in DM, all CAE engineers working in the particular project can be notified of the existence of a newer file of a part/subassembly, which is currently under work. In case of Minor Updates from CAD (which means that a new CAD Version is released for some of the parts) the translation Process will be executed only for the new parts. Queries to the DM, concerning the existence of new versions and study versions, can be executed through the “Check DM Updates” Tool.

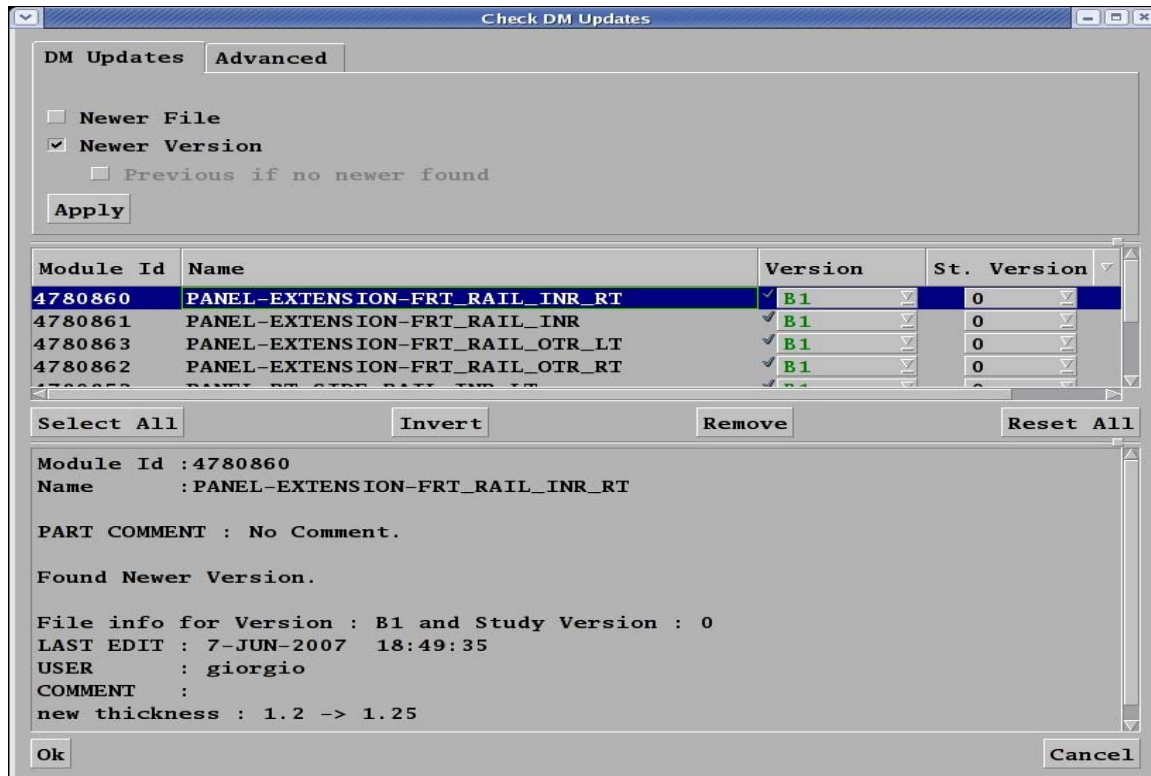


Figure 5

Parts of interest can be isolated and engineers can decide whether to import the new versions or not after having checked each part’s information and comments, as depicted in figure 5 above. All requested parts will be automatically integrated to the Models, while at the same time important information regarding the Connections, the Boundary Conditions and the Output Requests will be updated. It’s important to mention, that the Representations of the new parts can be created with the Representation Manager either before use or on a “just in time” request.

In the case of Major updates, where some of the previous parts have been deleted and new parts have been added in the model’s part list, the whole process will be repeated as previously described, starting with the translation of a new VPM-tree.

2. ANSA BATCH MESHING CAPABILITIES

2.1. The Batch Mesh tool

As mentioned above, when the Common Model's construction has finished, CAE engineers can continue working on the different FE Representations required to fully analyse the model/subassembly under consideration. The meshing tasks required for each Discipline are different and the recommended way to deal with them is through the Batch Mesh Manager.

Batch Mesh is the tool, available within ANSA, which performs automatic mesh generation on geometrical surfaces, (4). Its main characteristic is the ability to define several Batch Meshing Sessions, in order to mesh various parts of a model having set different meshing parameters. Two main sets of parameters are defined, when setting up a Batch Mesh scenario. The first one consists of the Mesh Parameters needed for the mesh generation, the several feature treatment tools as well as for quality improvement tasks. The second one contains all quality requirements and provides the ability to define threshold ranges. Batch Mesh also gives the ability to control a large number of features during mesh generation, such as dealing with fillets, flanges, holes, defeaturing and volumes, which will be explained in more detail in the paragraphs to follow.

At the end of the Batch Mesh procedure a report is generated, which presents general statistics regarding the mesh process and the quality achieved, based on the criteria and parameters that were set.

2.2. Batch Mesh scenario creation

For the purpose of demonstration, a vehicle model is considered, where Front, Side Rear and NVH analysis are requested. Figure 6 depicts such a model, where different colours are used to paint parts of the vehicle which belong to different scenarios. Obviously, several groups of parts need to be meshed using different parameters. In other words, a number of different mesh sessions must be introduced to meet the rules and requirements of the representation scenarios mentioned above. Through the Batch Mesh Manager tool, sessions can be fully customised and further subdivided to include sub sessions, where different meshing rules must be applied.

Consider the Front Impact case as shown in figure 7. The initial session has been partitioned according to each part's location in the vehicle. As a result three new sub sessions have been created, which are named the front, middle and rear section. The front section where the impact area is located, should have a small element size, the middle area a mid element size and at the rear a coarse one. This partitioning can be performed with the use of the ANSA Batch Mesh filters.

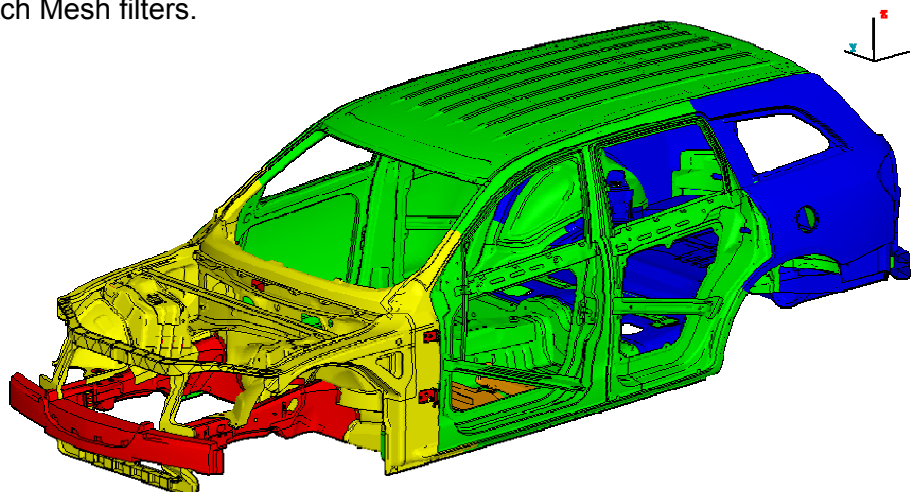


Figure 6

After the initial sectioning we need to identify some parts that need to follow different meshing rules from the rest. First of all we have to separate all the small parts of the structure. These parts need to be meshed with a finer element length and can be identified from their small area size, using the appropriate filter. At the next stage we have to identify all the important parts of the structure. These parts also need to be meshed with a finer element length from the rest of the area and can be recognized by a keyword in their name. In our example this way of filtering was applied for the front rails, the bumper and the cross members of the structure. Finally we have to separate the solid parts of the model that need to be meshed using volume elements.

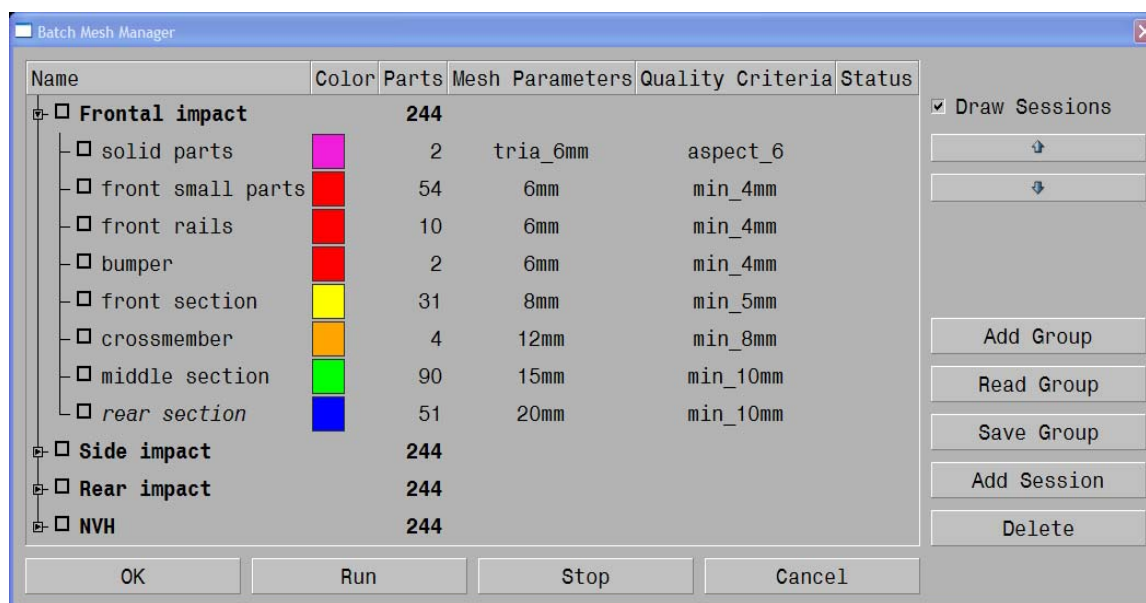


Figure 7

At this stage we have identified and partitioned our model in 8 different areas of interest. The next step is to define all the meshing rules for each area. These meshing rules consist of the mesh parameters and the quality criteria. Both of them can be set through Batch Mesh Manager. The meshing parameters are categorized in such a way that the meshing result of each session can be easily supervised.

2.3. Mesh Parameters

2.3.1 Shell Mesh properties

The first step in setting the mesh parameters is to define the shell mesh properties, depicted in figure 8. The available mesh types which can be selected from are Mixed, Quad, Tria, CFD and STL mesh, (4). Each of them has its own set of parameters so as to control the mesh size and the distortion rules of each session. The Distortion Distance is the maximum allowed normal distance between the nodal chord of a perimeter segment and the underlying CAD that defined it. Distortion Angle is the maximum allowed angle, measured in degrees, between the tangents of a perimeter segment at two consecutive nodes. The element order can also be defined, whereas an option is offered as to whether work should be done on existing mesh or any existing mesh should be erased before a new one is created. Options are also provided in order for triangular shell elements to be removed from Spots, Internal Single Bounds and Feature Lines.

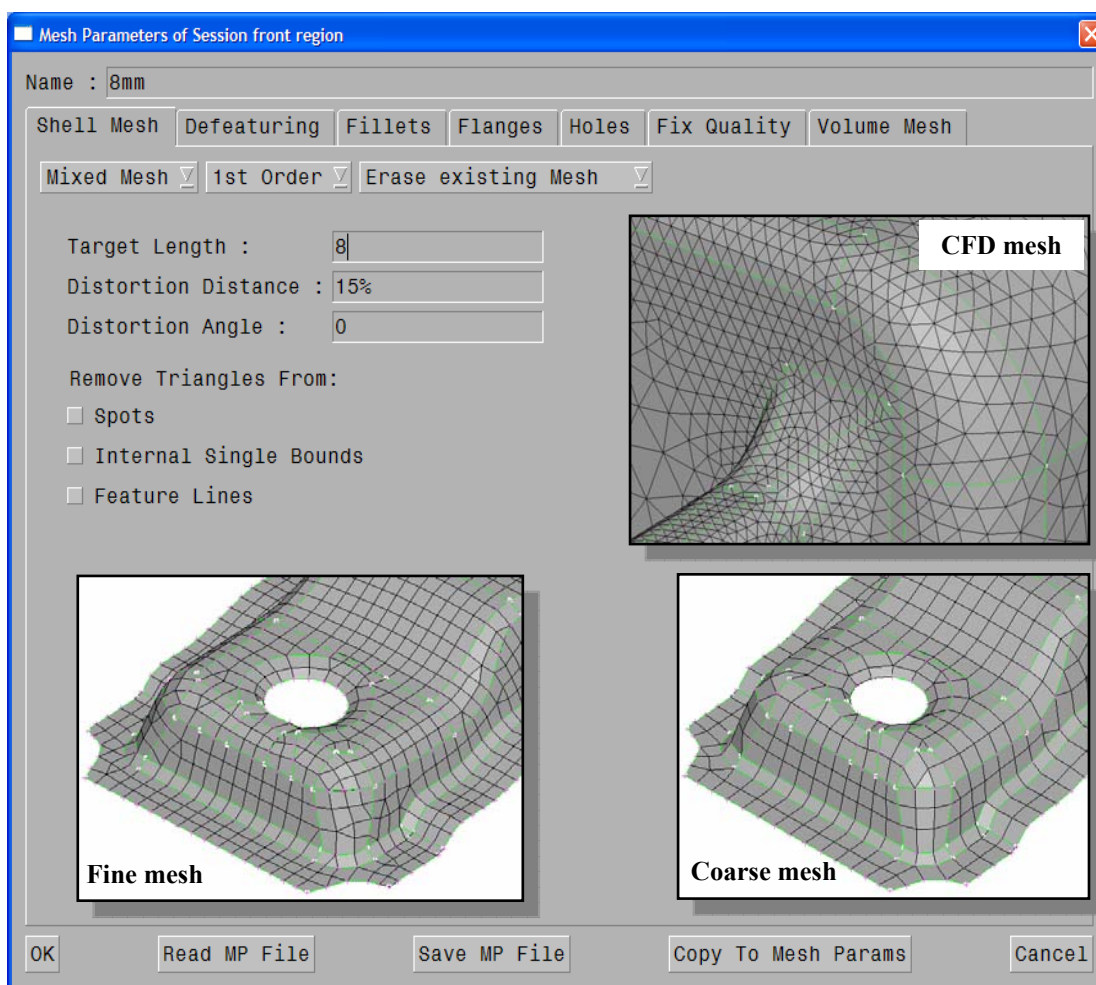


Figure 8

2.3.2 De-featuring

Moving on to the de-featuring section, (figure 9), in order to achieve the desired mesh result, several parameters can be set so as to define on which occasions perimeters should be joined or not and on which parts of the model Hot points should inserted or deleted. Feature Line Angles are very important as they determine which perimeters should be considered as important ones during the meshing procedure. The automatic identification of such perimeters is achieved through the use of the Feature Line Angle (4), which is defined as the supplementary of the angle formed between two adjacent surfaces on a common Perimeter.

Equally important is the use of the Feature Line's Corner Angle, in order to determine which corners between two Feature Lines are important. It is defined as the supplementary of the angle formed between two consecutive Feature Lines. The batch mesh tool also provides the ability to maintain characteristic perimeters during the meshing process, such as the symmetry plane and cuts which are used for model splitting.

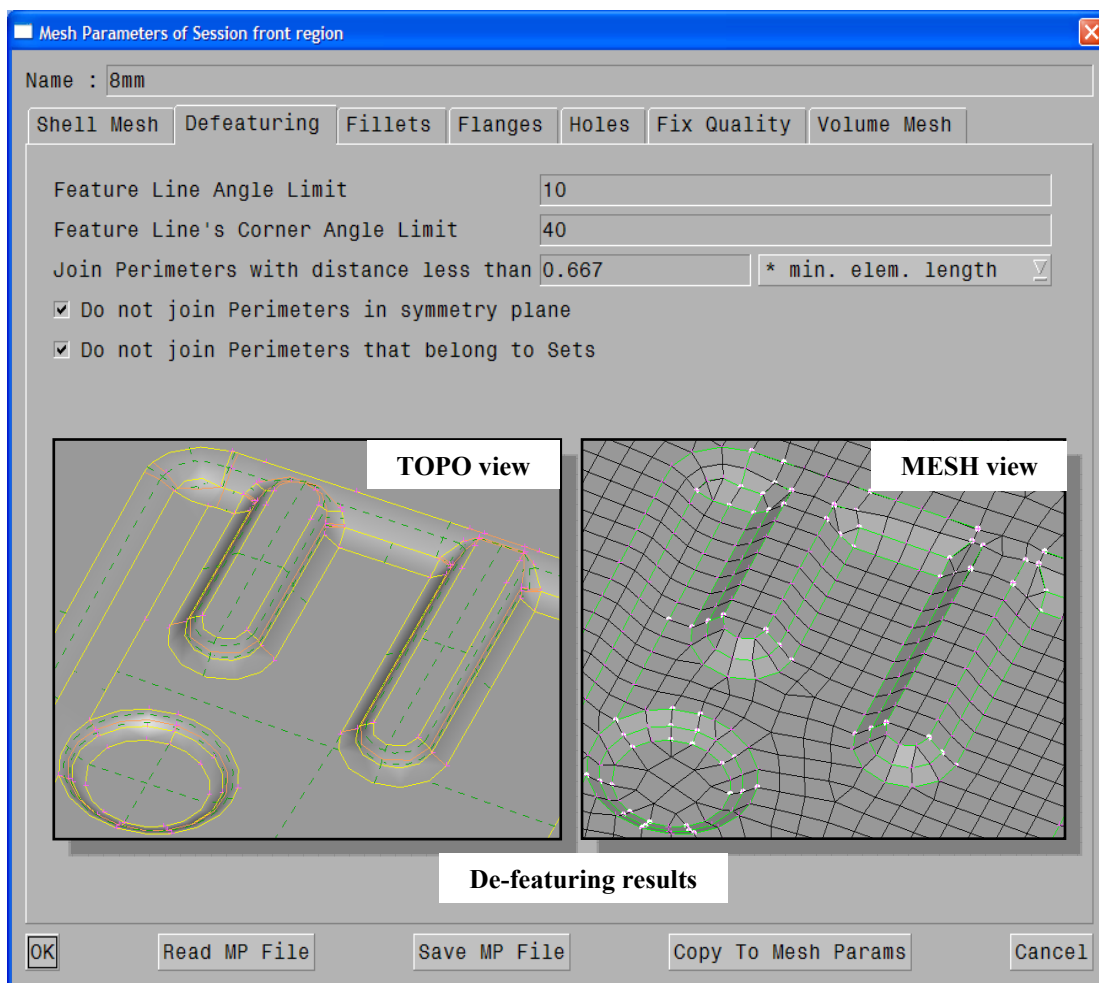


Figure 9

2.3.3. Fillet treatment

As far as fillets are concerned, their detection by the Batch Mesh tool, (figure 10), is based on their width and radius. The range of values is inputted, based on which they will be recognized and sets the desired method for treating them, (4). To be more specific, there are three ways available to deal with fillets, a) sharpening, b) splitting and c) creating element rows. The first one replaces fillets with the respective corner faces, while following the second one, fillets are cut in half iso-parametrically. Should the third way is selected all fillets will be meshed in order to create the requested rows of elements.

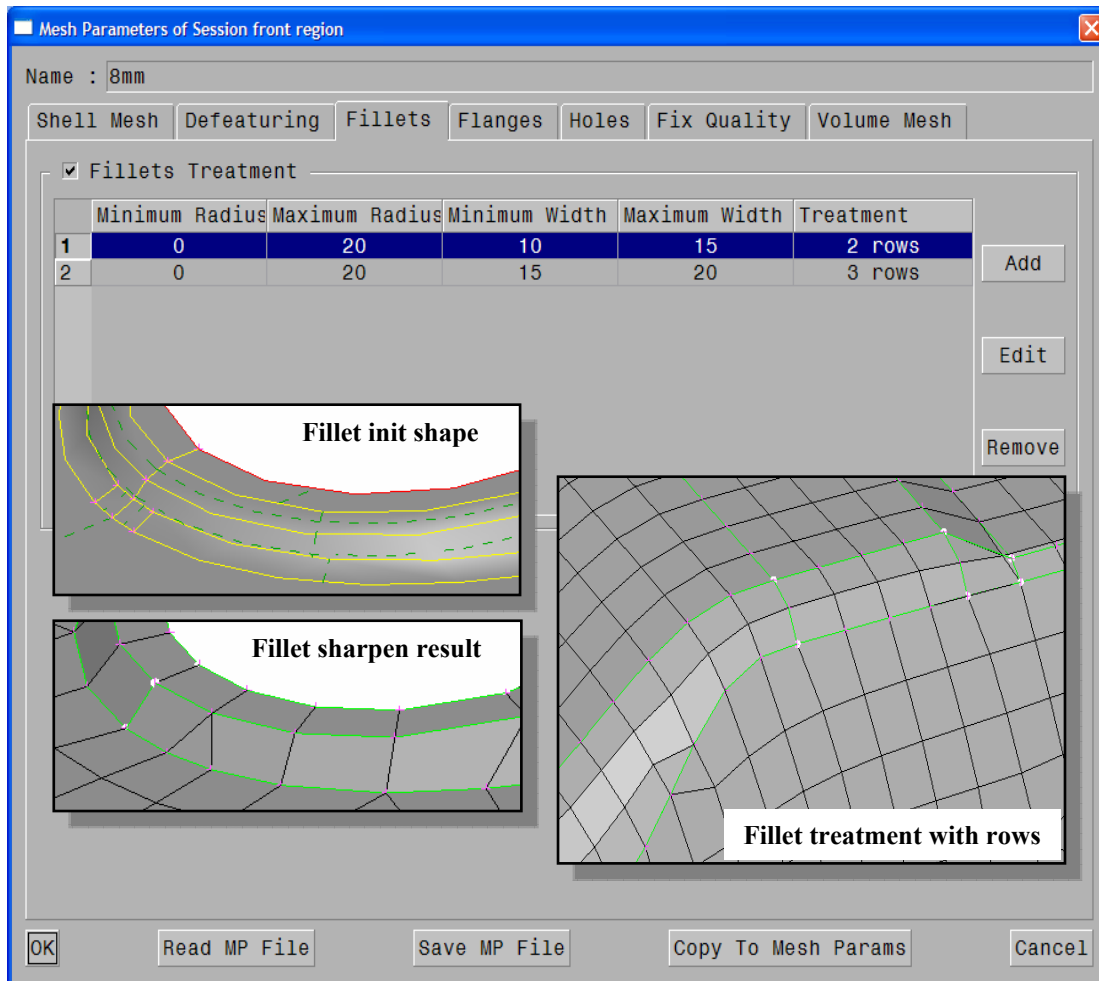


Figure 10

2.3.4. Flanges

Through the mesh parameters the treatment of flanges can also be set. The Batch Mesh tool, (figure 11), provides the ability to define the number of rows of elements that will be created during meshing. Flanges can be recognized in three different ways.

The first one is implemented with the use of faces proximity, (4). Flanges are detected by locating faces that lie opposite each other. Two variables are used, the distance and angle formed between normal vectors. The second way to recognize flanges is by studying the faces' shape. Faces that have at least one red bound and their width is in the range set in the flanges parameters section are identified. This algorithm is recommended in the case where each part is meshed separately and there are no connections available. Lastly, if there is connection data available, Batch Mesh can recognize faces close to connections based on the specified distance and identify flanges.

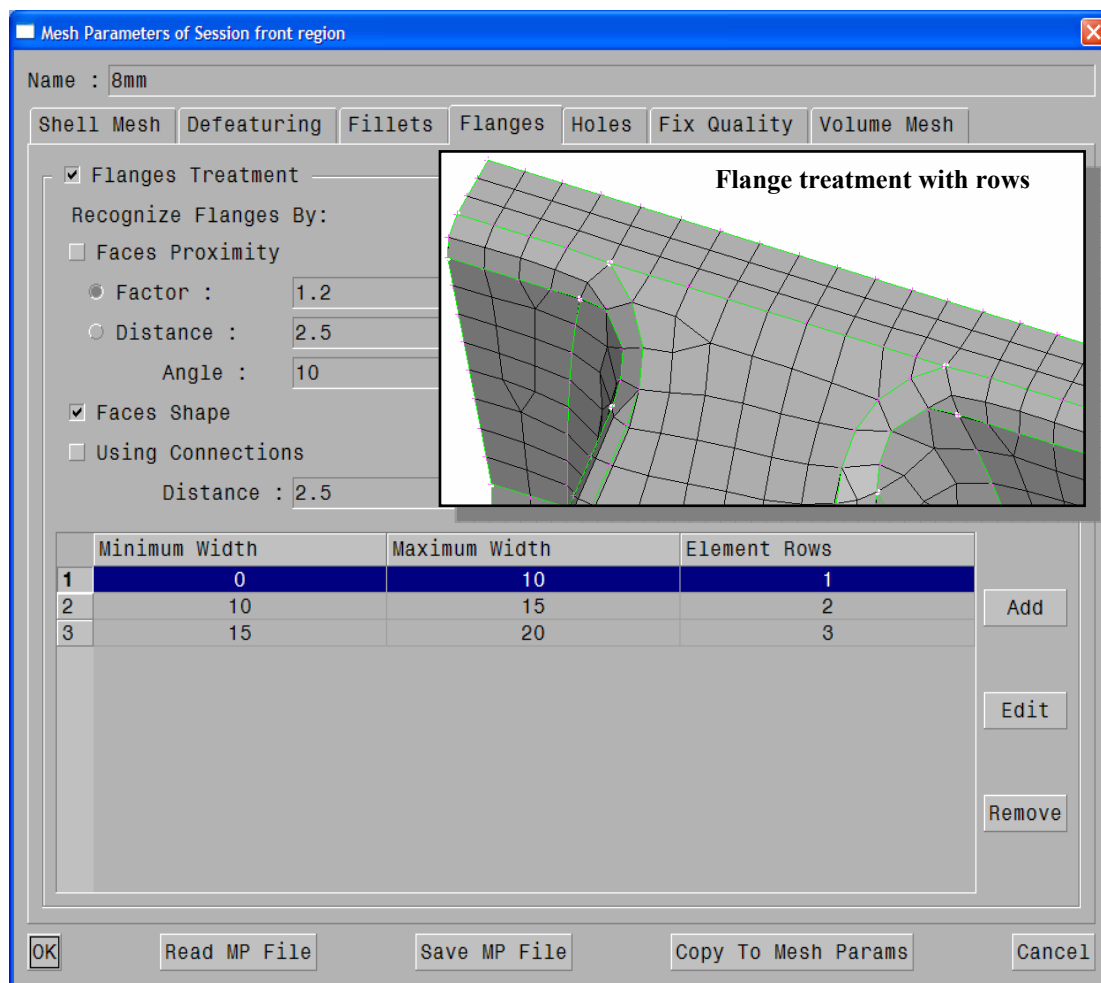


Figure 11

2.3.5. Holes treatment parameters

Batch Mesh also allows for the control of meshing generated around holes. Bolt holes parameters can be set separately from the rest and can be identified in three different ways in addition with the specified diameter range, as shown in figure 12. To begin with bolt holes are recognized by means of proximity, using the distance and angle formed between the normal vectors of the holes' centers, (4). Secondly, in the case connection information being available, the distance of the bolt connection to a center of a hole can be used to identify one. Lastly, all holes with specified shapes will be considered to be bolt holes.

Once holes have been recognized, certain options are given to set the holes' node number, fill holes and optionally mark the center point, resize the holes' diameter and/or created a specific number of zones around them. For holes falling within a certain diameter range a new diameter value can be defined. Additionally, the number of nodes in the holes' perimeter can be set, either as a stand alone number or given as a factor multiplied by the diameter. Last but not least, the number and width of quad zones, which will be introduced around holes, can be optionally defined.

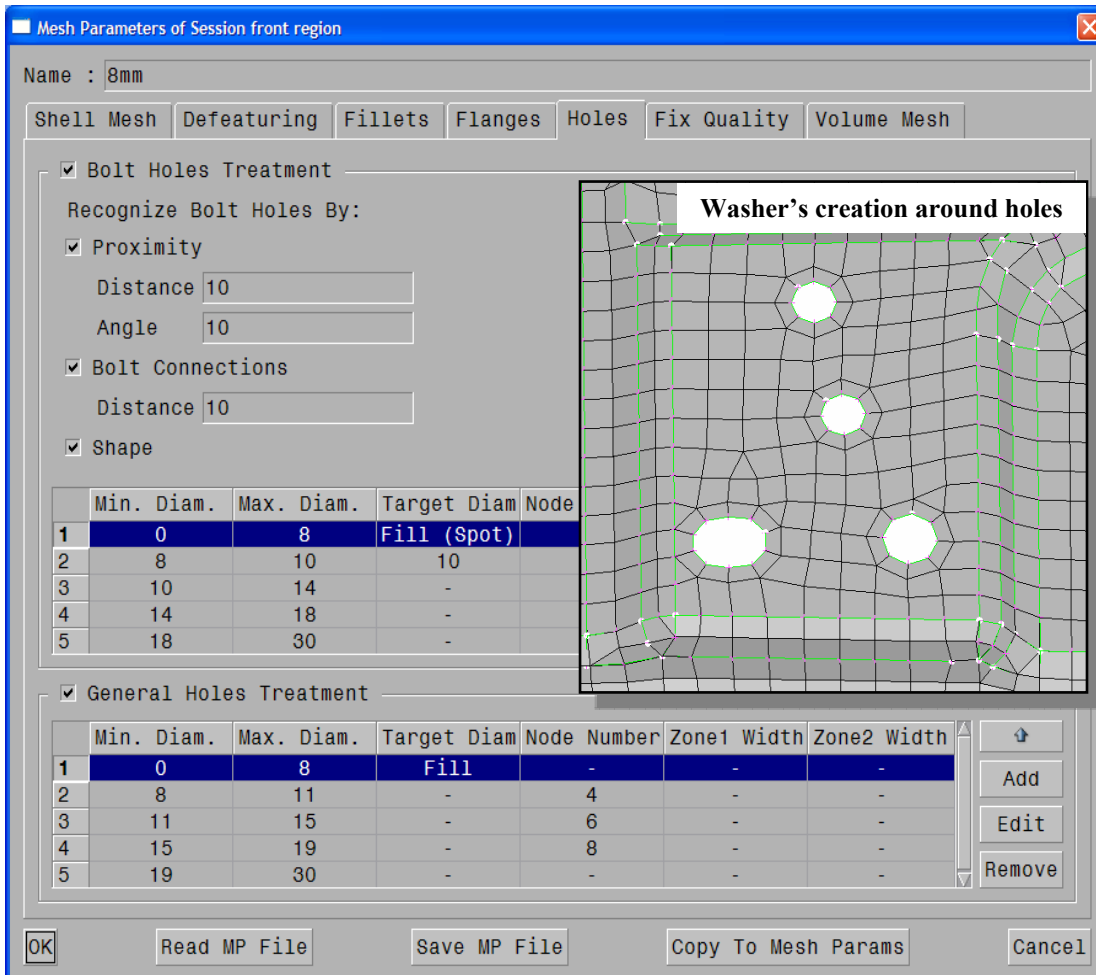


Figure 12

2.3.6 Fix Quality algorithm

Violating elements, which are generated as a result of the meshing process, can be dealt with a quality improvement algorithm that lies within the Batch Mesh tool. Fixes are applied according to quality criteria set, such as minimum element length, warping, skewness, taper and mid point deviation. Certain options are also included to specifically address quality violations and deal with them as requested, as depicted in figure 13 below.

First of all, a selection is offered as to whether the remaining violating quads, once the fix quality tool has been applied, should be split or not. Secondly, one can define whether minimum element length should be corrected on rows of elements created in fillets, (4). The same option exists for the case of element rows generated in flanges, where its correction is not always desirable. In this way, a larger number of rows can be requested than those allowed by the minimum length criterion. To continue an option is also given to allow for some small hole shape deformation in order to fix potential washer's creation problems. In order for violations to be dealt with, elements must be moved in several directions and distances, which are controlled constrains that define the maximum allowed movement from the surface or perimeter. Finally, it should be pointed out that all movements are performed as close to the original geometry as possible.

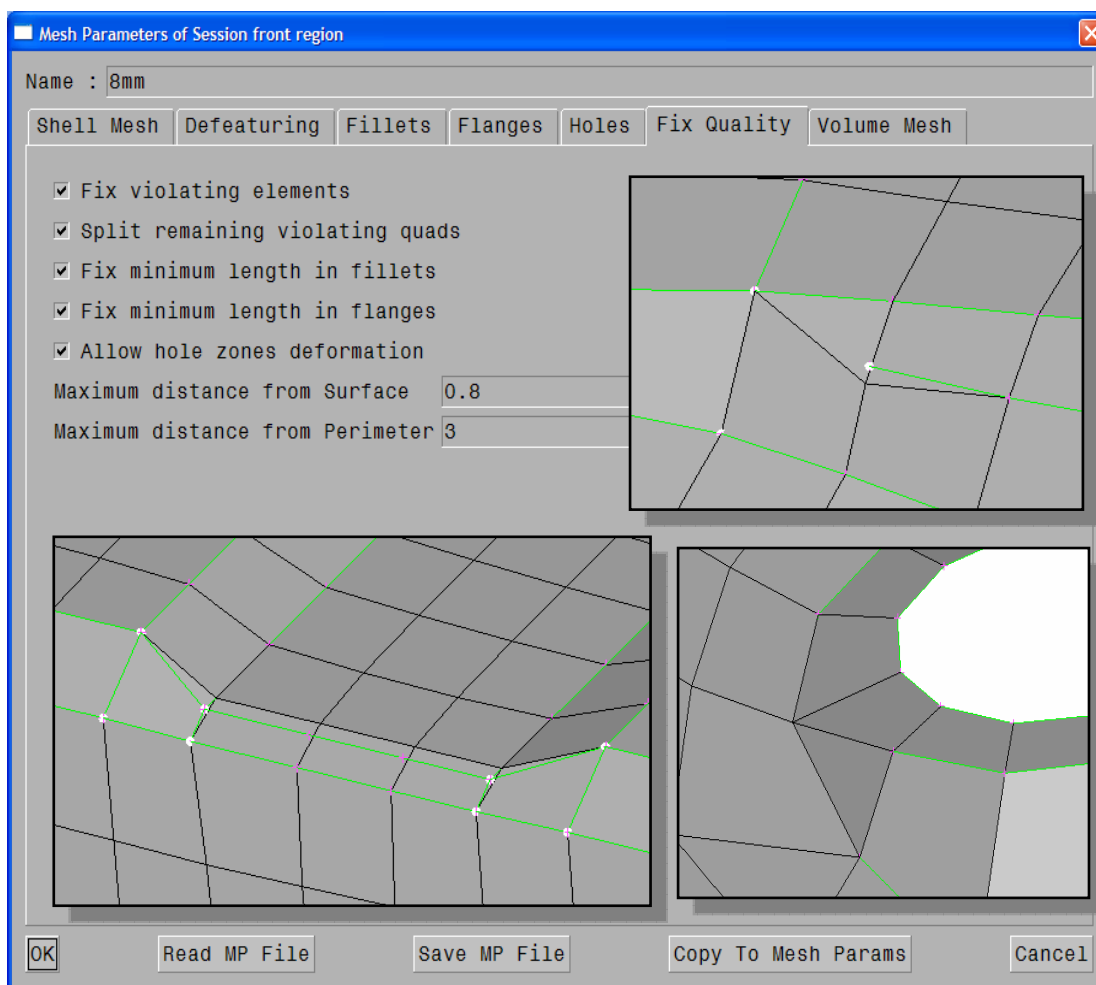


Figure 13

2.3.7 Volume meshing parameters

The last set of parameters to be set prior to batch meshing is the one regarding the meshing of closed volumes. Volume meshing can be achieved with the application of Tetra algorithm, while Layers and Hexa Interior creation are also supported, figure 14.

The Tetra algorithm is mainly suited for geometries of fairly thin volumes and nearly uniform surface element length, (4). As such it is applicable to most structural cases and some CFD scenarios. The generation of such elements is controlled by size and growth rate parameters. Layers support, allows for the creation of tetra elements which are surrounded by layers of hexas or pentas, which grow from all surfaces in a single step. Finally, with the set of Hexa Interior parameters a volume mesh can be created, which constitutes of a combination of hexa mesh inside the main volume and a transition zone to the surface mesh.

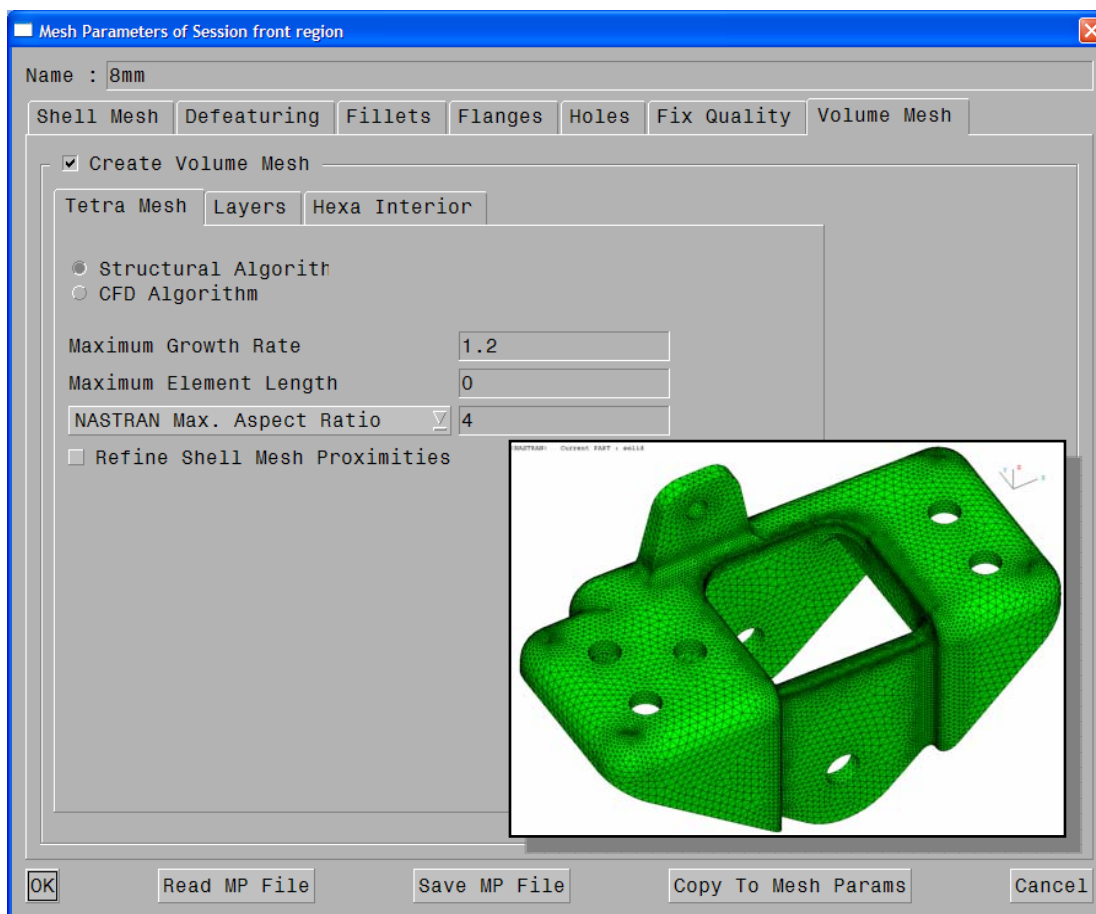


Figure 14

2.4 Mesh generation and further optimization

Once all mesh sessions have been defined for a specific Discipline scenario and all mesh parameters and quality requirements have been set, the Batch Mesh tool can continue with the requested mesh generation. Mesh parameters and quality criteria can be saved in ANSA DM so that they can be reused in different meshing scenarios where despite the fact that different sessions are to be executed the parameters to be set remain the same. As soon as the mesh generation has been completed, a statistics report is generated, figure 15, providing information on the results' status of each session that has been processed, (4).

Using the previously mentioned Batch Mesh Report Tool, the model's mesh quality can be thoroughly examined and any possible problems will be outlined. At the end it will be determined whether further improvement is required. Possible problems are considered to be quality violations and unmeshed macros. Once violating areas have been identified, fixing tasks can be scheduled to improve the meshed outcome. CAE engineers are then assigned different parts of the work to be done, which are handled in parallel so that the overall optimization process achieves the time saving and cost efficient goals that have been set at beginning of the project

After each part that has been checked and fixed is then published to the rest of the engineering group, simply by saving the representation in ANSA DM. The submitted part is then available for use by all potential meshing scenarios which might require the specific representation. The two most important aspects of the process described above, job subdivision and parallel working, are essential not only to meshing applications but also to other time consuming operations, such as the initial geometry clean up and are incorporated within the organizational capabilities of ANSA DM.

ANSA												
STATISTICS REPORT												
PARTS				STATISTICS			VIOLATING SHELL ELEMENTS					
Name	Module ID	Version	Session	Aver. Length	Unmeshed	Triangles %	Total	Aspect	Skewness	Warping	Crash	Min. Len.
CrossMember - LowerRad	60001	A1	front region	8.1429	0	1.21581	0	0	0	0	0	0
FRT-WHEELHOUSE_FRONT_RIGHT	60006	A1	front region	9.50528	0	8.98004	0	0	0	0	0	0
FRT-WHEELHOUSE_FRONT_LEFT	60005	A1	front region	9.46653	0	8.56354	0	0	0	0	0	0
EL COUPLER INNER LEFT	60004	A1	front region	8.42572	0	5.94724	23	0	4	17	0	10
LR JOINT LOAD BEAR OUTER LEFT	60002	A1	front region	8.3755	0	3.13117	7	0	0	5	0	2
R JOINT LOAD BEAR OUTER RIGHT	60003	A1	front region	8.39837	0	3.31633	4	0	0	2	0	2
TOTAL				8.58385	0	4.71725	34	0	4	24	0	14

Figure 15

3. CONCLUSIONS

Latest developments in the CAD process have made it very hard for the CAE engineers to keep up with the large number of model designs that are outputted by the design departments. Thus the overall meshing process, analysis and verification of the several component alternatives have become time consuming and error prone. Different model releases require the generation of several discipline scenarios, analysing the same or different model parts, with the use of a wide range of meshing parameters and quality criteria. At the same time it is extremely complex for CAE analysts to make sure that they work and compare results for the same models, since different alternatives may be used for the same part by different departments.

The proposed highly automated CAE workflow contributes to the robust developing of several discipline models, which follow specific guidelines and share common requirements. ANSA Data Management provides engineering teams with the latest model data versions, exhibiting effective communication methods and central organisation skills. DM drastically reduces developing time and cost, maintaining high process reliability, while at the same time allows for job subdivision and parallel working. Time consuming processes like geometry clean-up and meshing routines are performed only for newly arrived versions, while the joint use of the Representation Manager and Batch Meshing Tool guarantees that no task is performed twice, thus substantially reducing meshing time up to 90%. By implementing the suggested methodology CAE becomes faster and more flexible, allowing for better communications not only between CAE engineers working on the same project but also between the CAE and CAD departments, reducing the possibility of errors.

REFERENCES

- (1) BETA CAE Systems, "ANSA Task Manager Users Guide", ANSA v12.1.2 Documentation, 2007
- (2) Dimitrios Angelis, Stylianos Seitanis, "Efficient CAE DATA and process management, for virtual product development and verification, in LS –DYNA simulations", LS –DYNA Anwenderforum, Ulm 2006
- (3) BETA CAE Systems, "ANSA Data Management Users Guide", ANSA v12.1.2 Documentation, 2007
- (4) BETA CAE Systems, "ANSA Users Guide", ANSA v12.1.2 Documentation, 2007