Developing Failure Criteria for Application to Ship Structures Subjected To Explosive Blast Loadings

Mark Tyler-Street, Joke Luyten

TNO Defence, Security and Safety

Rijswijk, The Netherlands

Summary:

A research programme is being undertaken at TNO to investigate vulnerability reduction on warships. In this framework, studies have been performed regarding the structural damage due to both an internal missile explosion and close in explosions.

Due to the severity of the explosive loadings the structural deformation is considerable and an accurate method to predict both the initiation and progression of material damage would be of significant value to assist in the design of more blast resistant structures. The ability to model material failure is available with many of the material models in the explicit finite element code LS-DYNA although further development is required in order to correlate to observed experimental results.

In the material models, the initiation of failure is typically defined by the uni-axial failure strain e_f. This parameter is not an independent material constant and the failure characteristics will vary depending upon the applied stress state (for example sensitivity to tri-axiality), the temperature and rate of loading. For ship plate steel the manufacturing process of rolling may introduce anisotropic failure characteristics which differ in the rolling and transverse directions. The failure strain needs to be adjusted for coarser meshes which are unable to represent local strain gradients.

This paper describes some of the work that has been done to describe the failure properties of typical ship plate steel with the development of a user defined material model to predict and further understand the failure characteristics.

Keywords:

Blast loading, Dedicated Element, mesh size, failure strain.

1 Introduction

The transient blast loading from typical explosive threats (for example a missile strike or close in IED explosion) is considerable, resulting in significant structural deformation and failure. The ability to predict the extent of the deformation and failure is required in order to assist in the design of more blast resistant structures. At present it is not possible to predict the initiation and progression of failure reliably, as a result the objective is to predict the global deformation accurately and to include the failure process as best as possible.

Figure 1 Roofdier Trials internal explosion

Figure 2 USS Cole nearby external explosion on water surface

Due to the scale of the structures considered it is not possible to use a refined mesh capable of capturing the local strain gradients around the failure zone. Hence localisation limiters (for example non local methods) are not applicable in this case. With the coarse finite element mesh, the failure criteria needs to be adjusted for the larger elements in order to obtain the correct amount of energy dissipation and elongation before failure. Two methods are being developed to predict the failure of ship structures subjected to blast loading:

- Dedicated Element Methodology
- User Defined Material Model with Damage

The dedicated element has been developed specifically to predict the structural performance of the welded bulkhead to deck connections. The user defined material model is being developed to evaluate failure for the remaining structure and general applications.

2 Dedicated Element Methodology

The Dedicated Element methodology was developed by Dillingh at TNO [1] for the prediction of ductile failure of welded bulkhead-deck connections due to internal explosive air blast loading. It provide a means to predict the ductile failure of welded joints of ship bulkheads with the typical shell meshes used for ship analyses. This removes the need for substantial mesh refinement in the location of the welds and enables the large ship finite element models to be maintained at a reasonable size to enable acceptable solution times.

A ship bulkhead when subjected to a blast loading is strained by both bending and membrane straining. It is known that the ratio between the bending and membrane strain can have a significant effect on the failure of the welded connection between the bulkhead and deck and hence the total bulkhead energy absorbing capacities.

The dedicated element methodology represents the bulkhead to deck connection by a strip of shell elements along the weld line as shown in Figure 3. The strip is five element deep, with each local grouping of five elements along the length of the strip defining one dedicated element. This grouping of elements enables both the membrane and bending deformation to be measured over the group of elements for the calculation of the β ratio as shown in Figure 4.

The maximum plastic strain energy allowable in a dedicated element before failure as shown in Figure 5 depends upon the bending-membrane ratio β. The energy-beta relation was determined from detailed 2D finite element analyses [1] where the failure prediction is based on a critical combination of equivalent plastic strain and tri-axiality. When a dedicated element meets the bending-membrane ratio β failure criteria, it is removed from the simulation.

The dedicated element has been successfully implemented into LSDYNA as a user subroutine to evaluate blast loaded bulkheads as shown in Figure 6.

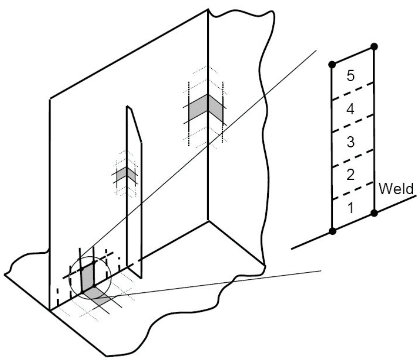


Figure 3 Definition of dedicated element in finite element model

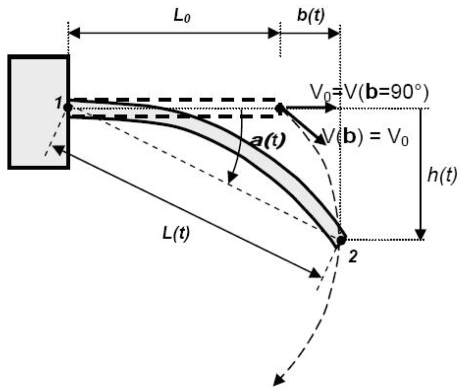


Figure 4 Definition of bending-membrane ratio β for dedicated element

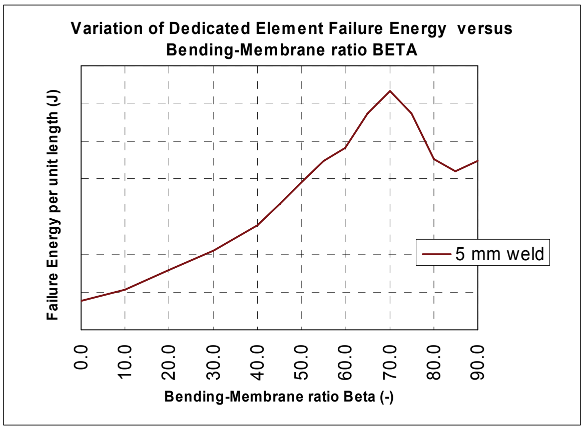


Figure 5 Variation of dedicated element failure energy versus Beta

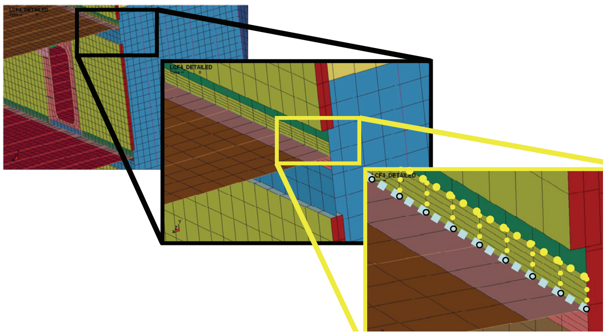


Figure 6 Implementation of dedicated element in finite element model

3 User Defined Material Model with Damage

The application of the dedicated element technology is limited to the simulation of the bulkhead to deck weld interface. For evaluation of failure of the remaining structure a user defined material model is being developed.

The user material model uses a standard plasticity model and incorporates modified functions for evaluating failure criteria. The strain at fracture is calculated similar to the Johnson Cook material model with functions describing the variation of failure strain due to stress triaxiality, strain rate and temperature. An additional function adjusts the failure strain with respect to element size automatically.

The failure criteria is calculated and stored for each individual element based on the element characteristic length to represent the adjustment of the failure criteria with respect to element size. This simplifies model creation as the mesh size is typically not uniform throughout a model and enables one material model to be used per part with different element sizes. An alternative is to group the elements in subsets of a similar size with an average failure property for that range of sizes. The adjusted failure strain for each element is calculated as:

$$\varepsilon_f = E(L_{cel})F(\sigma^*)G(\dot{\varepsilon})H(T) \quad (2)$$

In order to determine the variation of failure strain with element size a uni-axial tensile test was simulated with a range of different finite element sizes. In a uni-axial test the strain distribution is uniform along the gauge length before diffuse necking takes place with a subsequent increase in strains locally within the necked region as shown in Figure 7. Depending upon the gauge length employed different values of strain will be measured at failure as the measured value will include both the uniform and localised strain distribution.

In the simulations each mesh was loaded until the control gauge length reached the failure extension. This ensured an elongation consistent with experimental results for each mesh size and the variation in dissipated energy up to the failure elongation for the different meshes is negligible. The largest strain at the centre of the specimen was recorded at the failure elongation for each mesh size as shown in Figure 8. Similar variations in adjusted failure strain versus element length have been published previously [2].

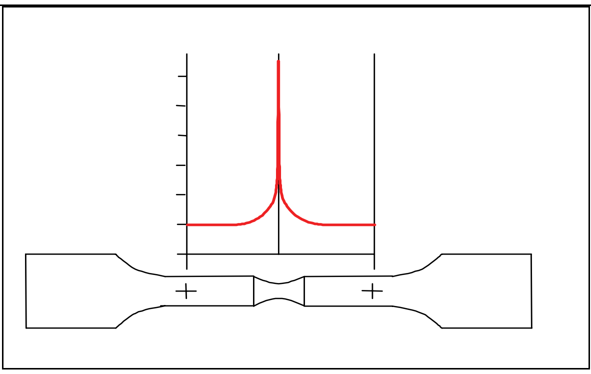


Figure 7 Variation in axial elongation and strain along gauge length

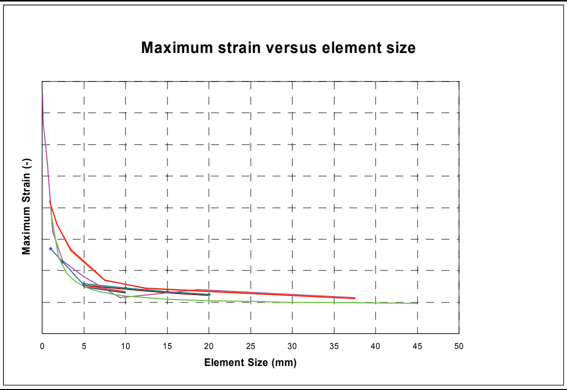


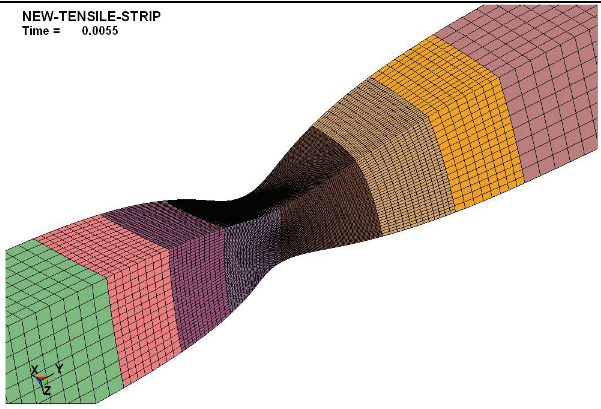
Figure 8 Variation in adjusted failure strain for different element lengths

The results show the variation of the average strain in an element over the failure zone with respect to element size. The curves are for the uni axial loading case where the triaxiality starts at 1/3 and increases to 2/3 before failure. For different loading cases, the magnitudes of the average strain would be expected to differ due to changes in loading history (for example effect of triaxiality).

The upper bound for the failure strain can be determined by the true fracture strain (essentially the fracture strain for a zero gauge length) by comparing the initial area A_0 and the final area after fracture A_f.

$$\varepsilon_f = \ln \frac{A_0}{A_f} \quad (3)$$

The failure properties were also investigated with respect to the rolling direction of the steel. The results of uni axial tensile tests for the different directions (rolling and transverse) showed ~ 2% difference in strain to failure (the strain measured over the gauge length). In contrast, the true fracture strain was ~ 50% lower in the transverse direction in comparison to the rolling direction. This was simulated with a series of different mesh sizes as shown in Figure 9 and Figure 10. The finest meshes were able to predict the neck reduction and difference in final area to match the experimental results.



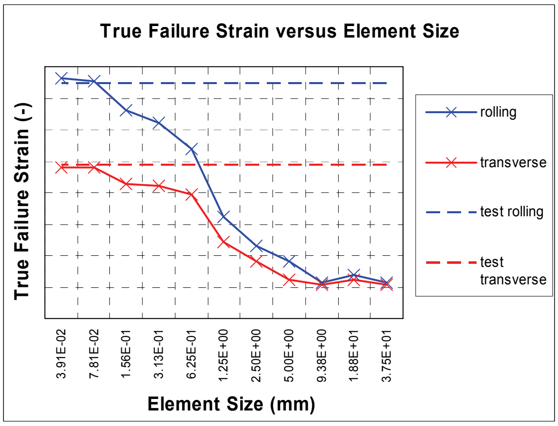
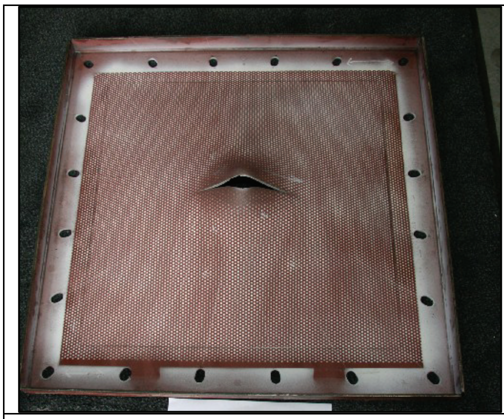


Figure 9 Prediction of true fracture strain (1/4 symmetric model)	Figure 10 Prediction of true fracture strain for different element sizes

The anisotropic failure properties may be seen in the results of unsupported plate tests as shown in Figure 11. In the experiment a single crack forms in the rolling direction, perpendicular to the weaker transverse direction. In a simulation using an existing isotropic failure criteria, cracks are formed in both the rolling and transverse directions. At present the failure criteria requires further work as it is still not possible to predict the anisotropic failure.



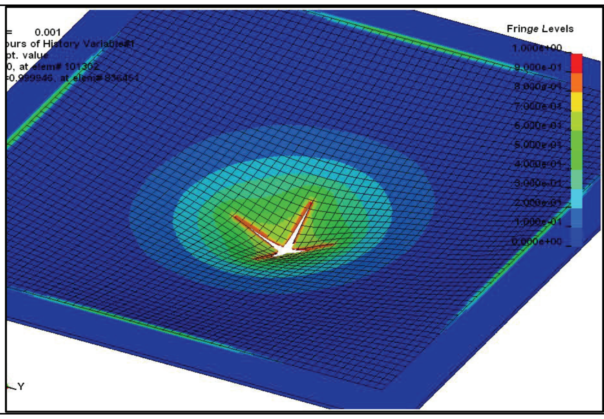


Figure 11 Close in explosion on unsupported panel with crack in rolling direction	Figure 12 Simulation of close in explosion with crack running in both rolling and transverse directions

4 Summary

The current status of the methodologies can be shown with two idealised models. The predicted failure of an bulkhead to deck weld is shown in Figure 13, only the innermost of the five elements (in a dedicated element group) is eroded from the calculation when failure occurs. In this case the β Ratio will vary along the length of the weld as the ratio of bending to membrane loading varies. With this method the correct failure load and energy dissipation per unit weld length were achieved for a relatively coarse mesh.

The results of a close in explosion at the water line are shown in Figure 14 for a simplified structure. The ALE method was used to model the explosive charge and surrounding air and water in order to apply the blast loading onto the structure. The anisotropic failure criteria is still under development, hence these results use an isotropic failure model.

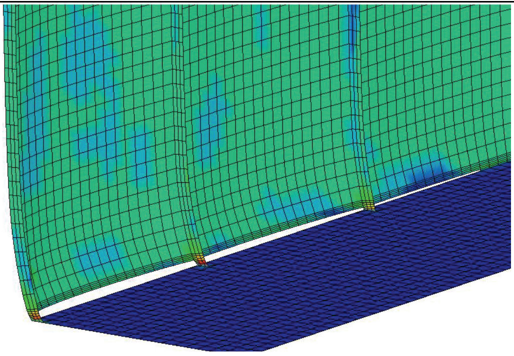
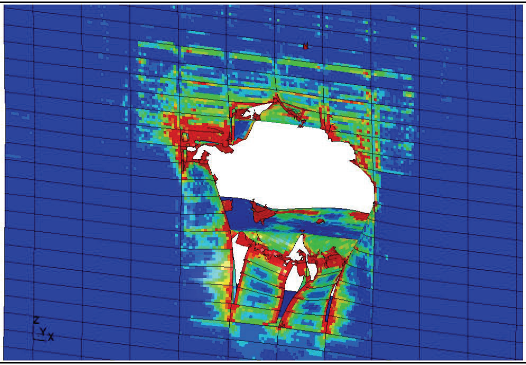
	

Figure 13 Dedicated Element simulation of a bulkhead to deck weld failure (simplified structure)	*Figure 14 Close in explosion on outer hull along waterline (simplified structure)*

With the current tools it is possible to model the effects of explosive loads on ship structures. Further improvements to the failure model are required in order to provide better predictions of the extent of both deformation and failure.

5 Literature

[1] Dillingh E.C. et al : "Implementation of a Dedicated Element for a weld under blast conditions into a finite element code", TNO Report, 2003.

[2] Florian Biehl: "Collision Safety Analysis of Offshore Wind Turbines", 4th LSDYNA European Conference , 2005.