ICSBE2016-241

FINITE ELEMENT SIMULATION OF THIN FOLDED MEMBRANES

B.Y.Dharmadasa* and H.M.Y.C. Mallikarachchi

University of Moratuwa, Sri Lanka
**E-Mail: yasara.dharmadasa@gmail.com, TP: +94776854118*

Abstract: Deployable gossamer structures use thin filmed membranes folded into different patterns. Predicting the membrane behaviour and the stress propagation in these structures is complex due to non-linear characteristics at fold-lines. This paper investigates some of the current idealization methods (perfect hinge, perfect weld) and proposes a novel method, where connectors with rotational elasticity are defined. Furthermore a thorough investigation on key factors affecting the output is carried out. The results have proved that the novel simulation approach proposed is more accurate and careful selection of key factors could make the simulation efficient.

Keywords: Deployable structures; Finite element simulation; Fold-line stiffness; Thin membranes

1. Introduction

Use of thin filmed membranes is becoming popular in gossamer space structures. Deployable structures can be stored at a compacted mode and later expanded to a larger dimension. Important factors when designing such structures would be the ease of deployment, stability of the deployed state and the level of compaction [1].

The thickness of a membrane is relatively negligible compared to other two dimensions. Therefore, a membrane covering a large surface area can be drastically compacted into a small compartment by folding it over and over again.

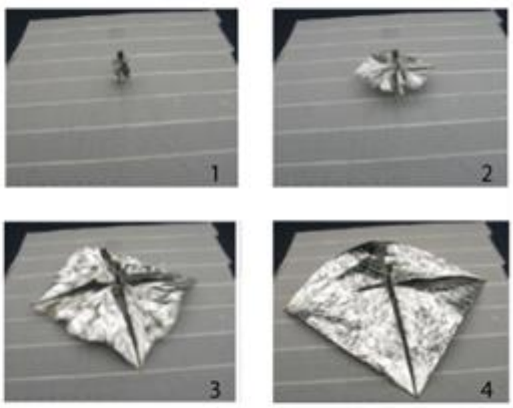


Fig 1 - Compaction of a solar sail using folding patterns [2]

Fig 1 illustrates the compaction of a solar sail used for deep space exploration missions [2].

Predicting the structural response plays an imperative role during the design phase. Use of computer programs has enabled engineers to analyse structures in virtual environments. Compared to physical testing, virtual simulations are efficient and economical. But the accuracy of the results depends on the method of idealization.

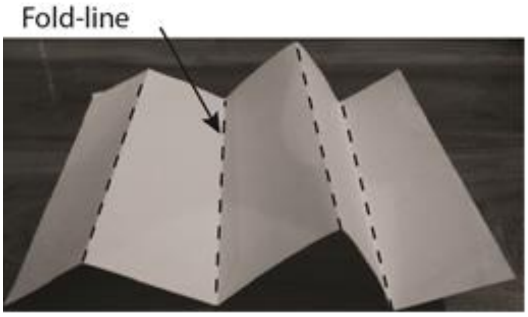
This paper is focused on the idealization of a fold-line in virtual simulations. We have attempted to understand the fold-line mechanics, and to develop a simulation technique that can accurately capture the membrane behaviour. The structure of the paper is as follows. Section 2 investigates the fold-line properties based on previous literature and section 3 suggests the finite element simulation technique. Section 4 analyse and discuss about the results obtained and section 5 concludes the findings.

2. Fold-line stiffness

When a thin membrane is folded along a line, a residual crease mark can be observed. This residual crease alters the material properties and creates a deviation in the membrane behaviour. This can be simply identified in Fig 2, where two identical papers are being observed after one folded and other unaltered. The geometric difference between the two is due to the effect of fold-line.

Microscopic investigations reveal the material deformation due to folding. Fig 4(a) illustrates the permanent kinked shape obtained due to a crease for 25 µm thick Kapton membrane. Creased paperboard, under a microscope, provides greater details [3] of the behaviour of fibre laminates (Fig 4(b)).

(a)



(b)

Fig 2 - (a) unaltered membrane & (b) folded membrane

It has been observed that tensile forces on top region have strained the fibres while compressive forces on bottom region have buckled them. These permanent deformations of fibres have resulted in a residual crease mark on the membrane. Next we will look at the mechanical behaviour of a crease. When folding loads are removed, the crease opens to some extent creating an arbitrary angle. This angle is identified as the "Neutral Angle" which dependents on factors such as material of membrane and level of scouring. When an external force is applied on the membrane to open or close the neutral angle, a resisting moment is generated. Lechanault et al.[4] has presented the relationship between opening angle and moment of resistance for 0.350 mm thick Mylar sheet (Fig 3)

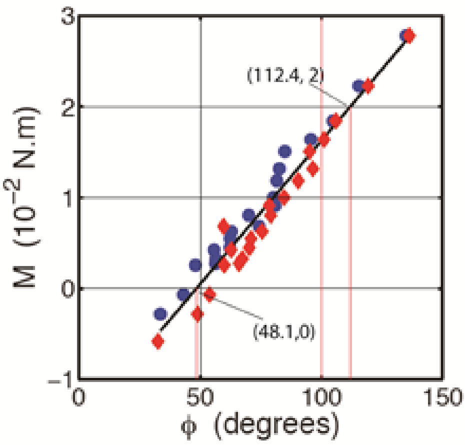
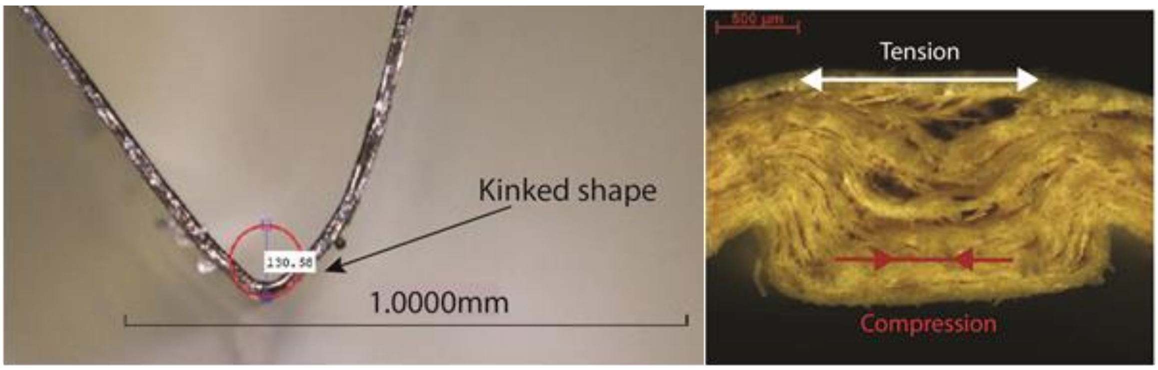


Fig 3 - Moment – angle relationship [4]



(a) (b)

Fig 4 - (a) Kapton polyimide crease & (b) Paperboard crease

According to the graph, creased membrane generates a linear resistive moment with regard to the opening angle. When the fold angle is 48° resistive moment is zero, which would correspond to the "Neutral Angle". Resistive moment (**M**) can be mathematically expressed using equation 1, where **θ** is the fold angle, **ϕ** is the neutral angle and **k** is the crease stiffness denoted by the gradient of the graph.

$$M = k(\theta - \phi) \qquad (1)$$

3. Finite element simulation

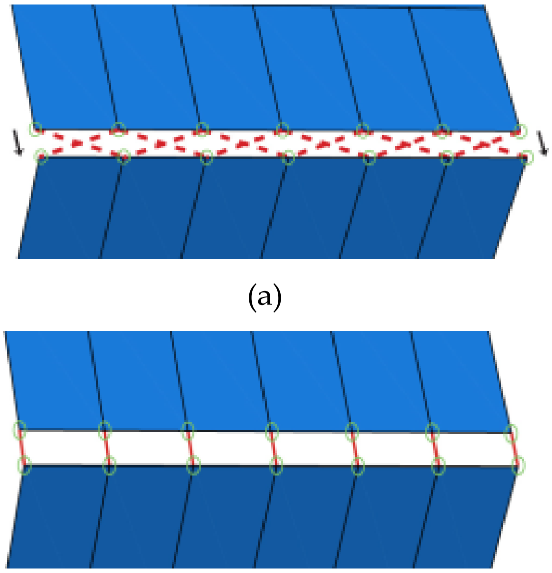
Using the above derived mathematical formulae, we have attempted to develop a finite element simulation for the simple experiment of 0.350 mm thick Mylar sheet. In this study the fold-line is idealized as a rotational spring, which is incorporated using connectors with rotational stiffness.

We have further looked into some of the other idealization methods in order to carry out a comprehensive study. Perfect hinge, which have been widely used in previous models [5], and perfect weld, which is the extreme opposite of perfect hinge are also considered. The simulation was carried out using Abaqus/FEA software package.

Finite element model was developed with two shell portions on either side of the fold-line connected at the common edge. Perfect hinge was connected using a tie constraint which locks the three translational degrees of freedom. In the case of perfect weld, three translational degrees of freedoms and rotational degree of freedom about the crease axis was constrained.

The novel approach brought forward in this paper lies in between perfect hinge and perfect weld. Two edges are connected using a tie constraint and additional connectors with a rotational elasticity are defined between two shells. When the fold is opening these connectors will be generating a resistive moment. The rotational elasticity is calculated by distributing the crease stiffness (k) by number of connectors per crease.

Two possible connector alignments are investigated in this study: connectors parallel to crease, and connectors perpendicular to crease. Prior to defining the connectors, two shells are moved apart in order to distinguish the nodes at the connecting edge. In the former method connectors are defined diagonally and the shells are moved back to original positions aligning the connectors parallel to the crease (Fig 5(a)). In the second method connectors are defined perpendicular to the crease and the shells cannot be moved back to their original positions since connectors length would disappear. Hence a gap of membrane thickness (0.35 mm) is kept between the two portions (Fig 5(b)).



(a)

(b)

Fig 5 - Connector alignment

(a) parallel to crease &
(b) perpendicular to crease

The simulation begins from completely folded state and a smooth displacement is applied on one edge while the other is pinned (Fig 6).

3.1 Element types

Having a proper understanding of the solver and the element types will be crucial to maintain the accuracy of results. Abaqus has two types of solvers inbuilt, implicit and explicit, which can be used for the analysis. The implicit solver is unconditionally stable, whereas explicit solver is only conditionally stable [6]. Even though our current study is

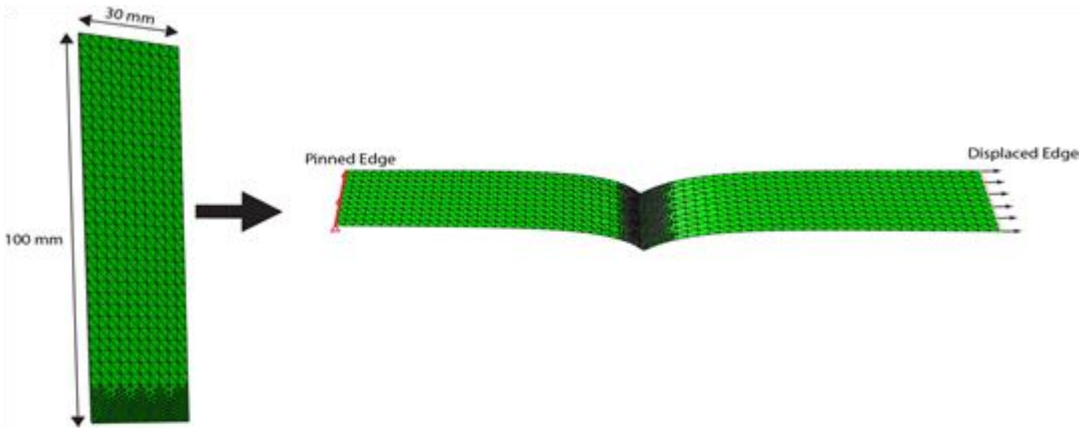


Fig 6 - Finite element model

relatively simple, real life applications (such as solar sails) would contain dynamic complexities which can only be solved using explicit solver. Therefore we have investigated the applicability of both solvers.

Anomalies in the energy diagrams can be used to identify the numerical errors that can occur in the explicit solver [7]. Shear locking and hour-glassing can cause irregular spikes in artificial and total energies of the model which would indicate possible errors within the simulation. Furthermore if quasi-static conditions are to be followed, kinematic energy should be negligible.

Table 1 indicates the types of elements we have considered in this study. We have looked at the accuracy and the computational cost of each element.

Table 1 - Element types

Element	Description
S4	4 node quadrilateral
S4R	S4 with reduced integration
S4R(h)	S4R with enhanced hourglass control
S4RSW	S4R with small strains considering warping effect
S3R	3 node triangular element with reduced integratuon
S3RS	S3R element with small strains

Sensitivity to number of connectors across a crease was carried out by varying the number of connecting points and their rotational stiffness as given in Table 2. Note that in each case the total stiffness of the crease is maintained at 17.82Nmm/rad.

Table 2 - Rotational elasticity

No of connecting points per crease	No of connectors per crease	Rotational elasticity (Nmm/rad)
2	2	8.910
3	4	4.455
4	8	2.227
5	5	3.564

4. Results and Discussion

4.1 Element Type

When selecting a suitable element type, accuracy and efficiency were given priority. As discussed earlier, energy diagrams were investigated to identify numerical errors. Elements S4R, S4R(h) and S4RSW had high artificial energies generated during analysis (Fig 7). Table 3 compares computational time and S4 element proves to be inefficient. It was identified that triangular elements are better suited for the task.

4.2 Deformed shapes

Deformed shapes of three idealization methods were compared with the experimental observations (Fig 8).

The perfect hinge mechanism is having a significant deviation from the experimental observation, and the membrane stresses are not captured in the finite element model. The perfect weld connection has achieved a better deformed shape but membrane stresses tend to be extreme. Resistive moment generated by the crease is decided based on material properties and cannot be altered.

Table 3 - Element performance

Element	Computational time
S4	>200%
S4R	46%
S4R(h)	34%
S4RSW	71%
S3R	100%
S3RS	54%

Connectors with rotational elasticity have produced most realistic output with almost exact deformed shape.

Also resistive moment generated is realistic and can be controlled with rotational elasticity.

4.3 Connector alignment and sensitivity

Sensitivity analysis on number of connector points has shown that few connectors tend to have high stress concentrations regions around the crease. But with the increment of connecting points, stress distribution tends to be uniform (Fig 9).

For a given number of connecting points, parallel alignment requires higher number of connectors. Furthermore, since it transfer stresses between two distinct points in the crease, connector points should be defined at small gaps. Therefore parallel alignment requires more connectors relative to perpendicular alignment thereby making it inefficient when building a large model.

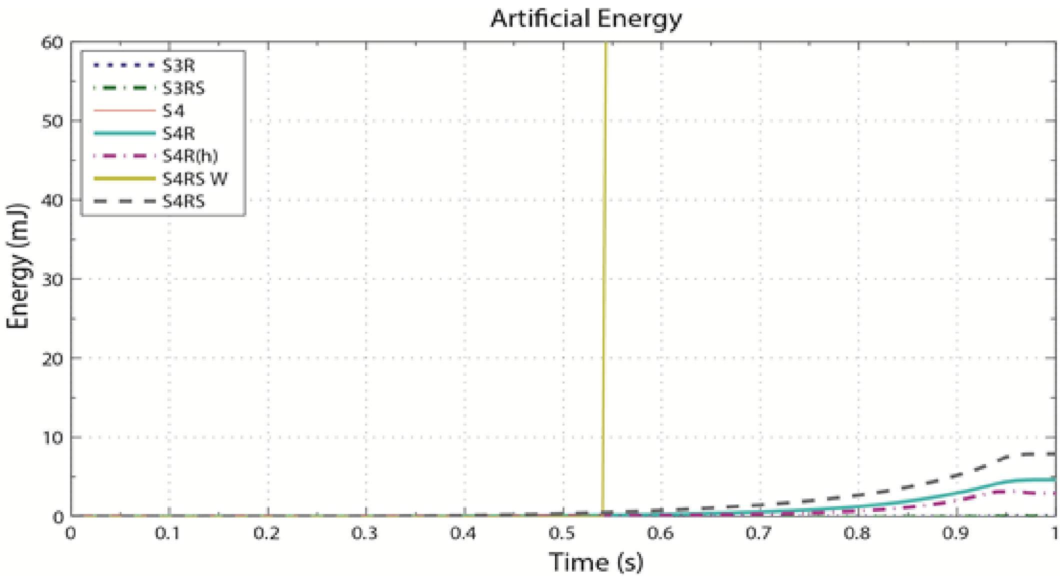
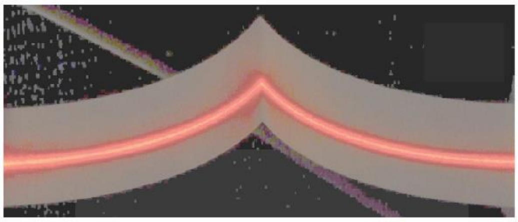
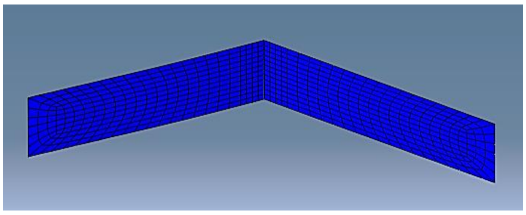


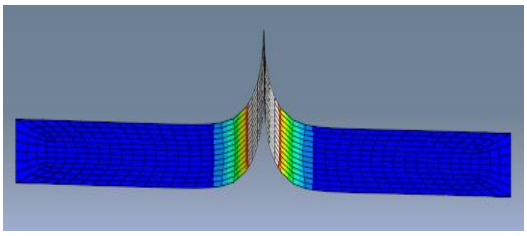
Fig 7 - Artificial Energy Diagram



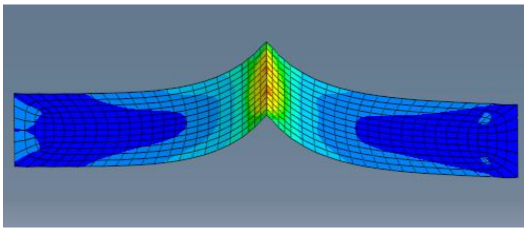
(a)



(b)



(c)



(d)

Fig 8 - Deformed shape of the crease (a) experimental results, (b) pinned connection, (c) fixed connection & (d) connectors with rotational elasticity

5. Conclusions and future works

Three different idealization methods for simulating a crease have been investigated and compared with an experimental observation. Connectors with rotational elasticity generated most realistic output that accurately captured the kinked shape at the fold. Also we have identified that perfect hinge crease, which is widely used in current simulations has few short comings. Failing to capture membrane stresses accurately is one such problem which could undermine the unfolding forces Importance of providing sufficient number of connectors was also highlighted in the study which would eliminate irregular stress concentrations. Analysis on element types suggests that S3R or S3RS elements are better suited for this simulation. Since linear elements are used in the analysis, crease regions should be carefully meshed in order to capture the curved shape. It is further recommended to carry out a mesh sensitivity analysis with respect to element type and curvature to improve the efficiency of the model.

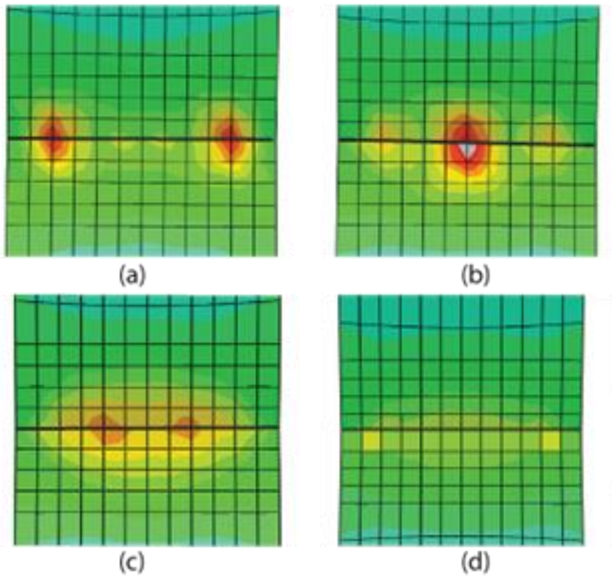


Fig 9 - Stress concentration at crease region

(a) 2 points (parallel)
(b) 3 points (parallel),
(c) 4 points (parallel)
(d) 5 points (perpendicular)

Acknowledgement

We thank Prof Sergio Pellegrino for helpful discussions, National Research Council of Sri Lanka and Senate Research Council of University of Moratuwa for providing financial assistance.

References

[1]. Guest, S. "Deployable structures: Concepts and analysis", Cambridge, 1994

[2]. Lappas, V. A. et al. "CubeSail: A low cost CubeSat based solar sail demonstration mission", in Advances in Space Research 48, 2011

[3]. Amigo, J. "Stiffness design of paperboard packages using the finite element method", Stockholm, 2012.

[4]. Lechenault, F., Thiria, B., & Adda-Bedia, M. "The mechanical response of a creased sheet", Physics review letters, 2014, p 112.

[5]. Liyanage, P., & Mallikarachchi, H. "Origami based folding patterns for compact deployable structures". 4th International Conference for

Structural Engineering and Construction Management. Kandy, Sri Lanka, 2013.

[6]. Abaqus 6.14, "Anaylsis User's Guide", Dassault Systemes Simulia Corp, Providence, Rhode Island, 2014

[7]. Belytschko, T., Liu, W., Moran, B., Bessa, M., & Elkhodary, K. "Nonlinear Finite Elements for Continua and Structures" (Second ed.). Wiley, 2013