A comparison of strain calculation using digital image correlation and finite element software

D Banerjee[1] and MA Iadicola[1]
[1]Material Measurement Laboratory, NIST, Gaithersburg, Maryland, USA
E-mail: Dilip.Banerjee@nist.gov

Abstract. Digital image correlation (DIC) data are being extensively used for many forming applications and for comparisons with finite element analysis (FEA) simulated results. The most challenging comparisons are often in the area of strain localizations just prior to material failure. While qualitative comparisons can be misleading, quantitative comparisons are difficult because of insufficient information about the type of strain output. In this work, strains computed from DIC displacements from a forming limit test are compared to those from three commercial FEA software. Differences between manually, DIC, and FEA calculated strains are assessed to determine if the scale of variations seen between FEA model and experimentally measured DIC strains constitute behavior differences or just numerical differences in the strain calculation methods used.

1. Introduction

Digital image correlation (DIC) data are being extensively used for many forming applications including constitutive law calibration, benchmark calibration, and for comparisons with finite element analysis (FEA) simulated results. The most challenging comparisons are often in the area of strain localizations just prior to material failure. This is because limit strains produce an inhomogeneous strain field prior to imminent failure. There is not a consensus on what constitutes “raw” data in DIC measurement and how estimates of errors and uncertainties in “raw” measurands (e.g., shape or displacement) affect the comparison of DIC to FEA. Smoothing related to DIC analysis parameters used for matching and strain calculation can affect the limit of spatial resolution and appropriateness of comparison to FEA [1]. For example, strain measurement uncertainty increases when the virtual gauge length (over which strain is ascertained) decreases [1]. While qualitative comparisons can be misleading, quantitative comparisons are difficult because of insufficient information about the type of strain output from FEA software. This is because FEA software often do not clearly explain how exactly strains are computed for a given type of element. Additionally, each software computes strain differently for a given type of element i.e. shell etc. In order to understand this, three benchmark problems are constructed, and the results obtained from three commercial FEA software are compared with manual computations (programed in GNU Octave) using linear shape functions and known displacements at nodes. Finally, strains computed from DIC displacements from a Marciniak [2] forming limit test (just prior to failure) are compared to those from three commercial FEA software. Quantitative differences in strains are assessed to determine if the scale of variations seen between FEA and DIC strains constitute behavior differences or just numerical differences in strain calculation methods used.

2. Overview of DIC system and strain calculation procedure

Although strains are frequently reported, the basic level of output from stereo digital image correlation measurements is an array of 3D initial shape positions (X,Y,Z) and the associated array of displacements (U,V,W) in that same coordinate system. In the measured Marciniak test reported here, a pair of 5MP CCD grey-scale lab grade cameras with 35 mm focal length lenses are used to acquire at 5 frames/s. The average magnification of the system is approximately 22.4 pixels/mm. Correlations were done over squares of 19 pixels (≈0.85 mm) with a raster step of 7 pixels (≈0.31 mm). The DIC manufacturer software [**3**] calculates strains using neighboring data points and applies a weighted smoothing with a diameter of five step points (≈ 1.25 mm here) centered on the point of interest decaying to a 10 % weight at the edges. Details of the software calculation are not known. To compare the DIC software and typical FEA calculations to manually calculated strains, the DIC data were organized into four node elements. For the manual calculations, the elements are defined in a local (ξ, η) coordinate system where linear shape functions are used to determine the displacement gradients in (ξ, η) space that are transformed back to physical space. Using these gradients, the Lagrangian strains are calculated including the second order terms at any points

Al-Sheikhly, Mohamad; Forster, Amanda; Gunnarsson, Carey; Jenket II, Don; Paulter Jr., Nicholas; Weerasooriya, Tusit.
"An Investigation of the Temperature and Strain-Rate Effects on Strain-to-Failure of UHMWPE Fibers."
Paper presented at Society for Experimental Mechanics 2016 Annual Meeting, Orlando, FL. June 6, 2016 - June 9, 2016.

of interest in the element. From these values the principal strains and true Hencky strains (ε_{xx}, ε_{yy}, ε_{xy}) are calculated at each point of interest. This formulation of strain does not include the effects of an element that is not initially flat. Although the specimen is nominally flat to begin the test, the measured values do not exactly match a flat X-Y plane. To correct for this each element was rotated and unfolded to flat as part of the manual strain calculation. This flattening procedure held the length and relative orientation of two edges and one diagonal of the element constant resulting in some induced small strains in the calculation, on the order of 10^{-6} strain. Calculations for strains at the integration points were done on each individual element using the lower left node as the reference point for rotation and unfolding to flat. Strain at each node was calculated separately on each of the four elements that neighbor that node (after the afore mentioned element rotation), and then the strains were averaged at that point weighted by the area of each contributing element. Missing elements due to lack of data at a node point were not included in the average calculation.

3. **FEA analysis overview and software used in this study – description of elements used**

Three different problems were considered: (A) a 2-element uniaxial (strictly planar) deformation problem, (B) a 2-element by 2-element patch from the high strain band (see Fig. 3), and (C) the entire area near the high strain band at about 0.4 s prior to fracture in Marciniak test (see Fig. 3). FEA study was conducted using three commercially available software: ABAQUS, ANSYS, and LS-Dyna[i] [4-6]. Based on our selection of problems two different types of elements were used, either 4-noded plane strain or 4-noded shell/membrane element. For Problem A plane strain elements were used since there was no variation in Z-coordinate among the nodes and the strains were functions of planar coordinates alone and the out-of-plane normal and shear strains are equal to zero. Plane strain elements are defined in the X–Y plane, in which all loading and deformation occur. Shell elements are used to model structures in which dimension in one direction is significantly smaller than the other dimensions. Typically, shell elements use this condition to discretize a domain by defining the geometry at a reference surface, where the thickness is defined through the section property definition. Traditional shell elements have displacement and rotational degrees of freedom. Membrane elements (and shell elements with a membrane option) are used for Problems B & C. These elements are akin to shell elements and are essentially surface elements that transmit in-plane forces only (no moments) with no bending stiffness. Element formulations vary among commercial software. In this work, efforts were made to use comparable elements in FEA software to solve the three problems. The elements used are: ANSYS (PLANE182 and SHELL181 with membrane option), ABAQUS (continuum plane strain CPE4 and membrane M3D4), and LS-Dyna (Shell element with plane strain and fully integrated option). Note the following pertaining to the FEA calculation: (a) although both reduced and fully integrated elements were used, only results from fully integrated elements are reported here, (b) FEA comparisons with DIC were done at both node and integration points, (c) non-linear geometry option was always turned on, (d) only linear shape function elements were used, and (e) both Lagrangian and Hencky (true) strains were computed but only Hencky strains are reported for brevity.

4. **Benchmark problems and discussion of results**

Problem A, the 2-element model (N_1-N_2-N_4-N_3 and N_4-N_6-N_5-N_3), was constructed using quadrilateral elements (Fig. 1). The left 2 nodes (N_1 and N_2) were constrained in X, and Y directions. A 0.1 mm displacement in the X-direction was applied to nodes N_3 and N_4 and a 0.3 mm displacement in the X-direction was applied to nodes N_5 and N_6. ANSYS Plane182, ABAQUS CPE4 and M3D4, and LS-Dyna Shell (with plane strain option) elements were used to compute the true normal and shear strains at integration points. Note that each fully integrated elements [7] has 4 integration points that are numbered starting from lower left corner and are incremented in counterclockwise manner in each element. Table 1 lists ε_{xx} computed using the procedure described in Section 2 above (hereafter called "manual calculation") and those computed using the three commercial FEA software. The differences between strains obtained with manual calculation and from each software are listed under the "Difference column". Similar results are obtained for the ε_{yy}, and ε_{xy} but are not shown for brevity. It is clear from this table that although all FEA software produce reasonable values, ABAQUS membrane element is the most consistent with the manual calculation. A similar exercise was conducted on a 2 element by 2 element patch from the

Marciniak test just before onset of failure (Fig. 3), Problem B. This patch is shown in Fig. 2. For this exercise, the displacement values at each node were used as boundary conditions in a simple static analysis. Strains computed at each integration point were compared to those computed manually. In addition,

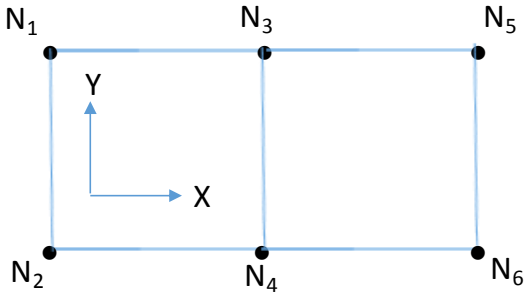

Fig. 1 Problem A.

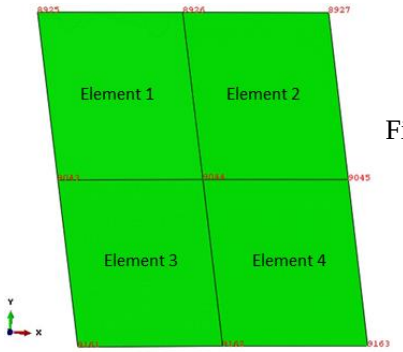

Fig. 2 Problem B.

average nodal strains computed at the common node in the center of the patch are compared. Table 2 shows the true strain values at the integration points. Only results from element 4 are shown here. Again,

Table 1 Problem A ε_{xx} at integration points

	int	Manual	ABAQUS - CPE4		ABAQUS-M3D4		ANSYS-Plane182		LS-Dyna-full integration	
	pnts	calculation	Value	Difference	Value	Difference	Value	Difference	Value	Difference
Element 1	1	0.095348	0.095269	-7.90E-05	0.095349	6.08E-07	0.095142	-2.07E-04	0.095369	1.70E-05
	2	0.095348	0.095269	-7.90E-05	0.095349	8.17E-09	0.095142	-2.07E-04	0.095234	-1.18E-04
	4	0.095349	0.095268	-8.10E-05	0.095349	-3.15E-08	0.095141	-2.08E-04	0.095382	2.90E-05
	3	0.095349	0.095268	-8.10E-05	0.095349	-6.31E-07	0.095141	-2.08E-04	0.095253	-1.00E-04
Element 2	1	0.182323	0.182319	-4.00E-06	0.182323	-4.61E-07	0.181814	-5.09E-04	0.182340	1.70E-05
	2	0.182323	0.182319	-4.00E-06	0.182323	-4.61E-07	0.181814	-5.09E-04	0.182290	-3.30E-05
	4	0.182323	0.182319	-4.00E-06	0.182323	-4.68E-07	0.181814	-5.09E-04	0.182340	1.70E-05
	3	0.182323	0.182319	-4.00E-06	0.182323	-4.68E-07	0.181814	-5.09E-04	0.182310	-1.30E-05

ABAQUS with membrane elements provide the best match to the manual calculation. Both LS-Dyna and ANSYS values are similar and the difference between manual calculation results and values predicted by these software are on the order of 10^{-3}. Note that ε_{xy} are tensor shears in manual calculation which is half the value in vector or Voigt notation in FEA. Table 3 shows the results at the central node, 9044. It is clear that ABAQUS predicted nodal values are closest to manual calculation values. ANSYS predicts slightly larger differences and predicts somewhat better values than LS-Dyna. There are many assumptions in the averaging technique and it is not clear how the nodal strain values are averaged by each commercial FEA software. Finally, the nodal results are compared in Fig. 3 for the actual forming limit test just prior to failure. Here, manual average ε_{xx} are shown in the upper portion of Fig. 3, while the bottom portion shows the difference between this calculation with DIC software (smooth), ANSYS membrane 181, LS-Dyna shell, and ABAQUS membrane elements. In this scale of +/- 0.02 strain difference, the DIC smoothing

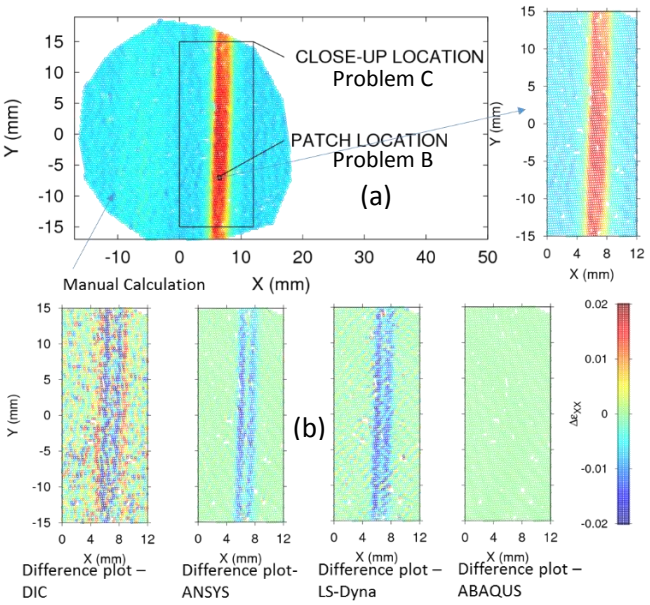

Fig. 3. (a) Manual ε_{xx} and (b) manual and FEA computed ε_{xx} difference plots in a high strain band prior to fracture in a Marciniak test.

option shows somewhat large strain difference from those obtained with manual calculation (with large differences along the periphery of the band and some random differences over the entire domain). While FEA software used shows better agreement with manual calculation in general (excepting near the high strain band), ABAQUS membrane element provides the best match with the manual calculation. Both ANSYS and LS-Dyna show similar behavior and show strain up to 0.02 strain below the manual calculation at the strain band, which is a somewhat large difference. The present exercise makes the point that reporting of strain values should be made with a better description of how the strains are calculated, what types of elements (along with integration scheme) are used, and what type of averaging scheme is used for obtaining nodal values from element integration point values.

Table 2. Problem B results at integration points.

Element 4		**Manual calculation**		**ANSYS-mem181-full int**			**Difference**		
Int Pt	**ε_{xx}**	**e_{yy}**	**e_{xy}**	**ε_{xx}**	**e_{yy}**	**e_{xy}**	**e_{xx}**	**e_{yy}**	**e_{xy}**
1	0.361250	-0.023102	-0.001579	0.36577	-2.11E-02	2.66E-03	4.52E-03	1.97E-03	5.82E-03
2	0.361250	-0.006998	-0.001845	0.3554747	-5.94E-03	-5.19E-03	-5.78E-03	1.06E-03	-1.51E-03
3	0.361330	-0.004073	0.004426	0.3557066	-6.08E-03	8.29E-03	-5.61E-03	-2.01E-03	-5.60E-04
4	0.361320	-0.020195	0.004531	0.3659433	-2.12E-02	1.64E-02	4.61E-03	-1.03E-03	7.35E-03
				ABAQUS-M3D4-full int			**Difference**		
1	0.361250	-0.023102	-0.001579	0.361250	-0.023102	-0.003144	0.00E+00	2.00E-07	1.37E-05
2	0.361250	-0.006998	-0.001845	0.361253	-0.006998	-0.003672	3.00E-06	-1.00E-08	1.76E-05
3	0.361330	-0.020195	0.004531	0.361335	-0.020195	0.009076	5.00E-06	3.00E-07	1.42E-05
4	0.361320	-0.004073	0.004426	0.361325	-0.004073	0.008869	5.00E-06	2.00E-08	1.71E-05
				LS-Dyna-membrane-full int			**Difference**		
1	0.361250	-0.023102	-0.001579	0.354880	-0.016713	-0.039607	-6.37E-03	6.39E-03	-3.64E-02
2	0.361250	-0.006998	-0.001845	0.350370	-0.001411	-0.041182	-1.09E-02	5.59E-03	-3.75E-02
3	0.361330	-0.020195	0.004531	0.351100	-0.001625	-0.041257	-1.02E-02	1.86E-02	-5.03E-02
4	0.361320	-0.004073	0.004426	0.355600	-0.016928	-0.039682	-5.72E-03	-1.29E-02	-4.85E-02

Table3. Problem B results at central node (9044).

Nodal strains at the central node (9044)							
	Manual	ANSYS-membrane 181		ABAQUS-M3D4		LS-Dyna-membrane	
Strains	Value	Value	Difference	Value	Difference	Value	Difference
ε_{xx}	0.367110	0.367199	-8.948E-05	0.367076	3.35E-05	0.360025	7.08E-03
ε_{yy}	-0.025112	-0.024335	-7.77E-04	-0.025218	1.05E-04	-0.020309	-4.80E-03
ε_{xy}	-0.008190	-0.012267	-4.11E-03	-0.016534	1.54E-04	0.000331	-1.67E-02

5. **References**

[1] Iadicola M A and Creuziger A 2015 *Uncertainties of Digital Image Correlation Near Strain Localizations* Proc. of the 2014 Annual Conf. on Experimental and Applied Mechanics (SEM)

[2] ISO 2008 *Metallic materials-Sheet and strip-Determination of forming-limit curves-Part 2: Determination of forming-limit curves in the laboratory* ISO12004-2:2008(E)

[3] VIC3D (v2010) http://www.correlatedsolutions.com/.

[4] ABAQUS 13.6 software Dassault Systemes http://www.3ds.com/.

[5] ANSYS Mechanical Release12.1 ANSYS Inc. http://www.ansys.com/.

[6] LS-Dyna-971 Livermore Software Technology Corporation http://www.lstc.com/.

[7] Zienkiewicz O C, Taylor R L and Zhu J Z 2013 *The Finite Element Method: Its Basis and Fundamentals* 7th Edition (Oxford: Butterworth-Heinemann)

[i] Certain commercial software or materials are identified to describe a procedure or concept adequately. Such identification is not intended to imply recommendation, endorsement, or implication by NIST that the software or materials are necessarily the best available for the purpose.