Using Computational Simulation for Analysis and Development of Next-generation Orthopaedic Devices

Emer M. Feerick, PhD
J. Patrick McGarry, PhD

Topics from the frontiers of basic research presented by the Orthopaedic Research Society.

From the Department of Mechanical and Biomedical Engineering, College of Engineering and Informatics, National University of Ireland Galway, Galway, Ireland.

Dr. Feerick is a member of a speakers' bureau or has made paid presentations on behalf of Invibio Biomaterial Solutions. Dr. McGarry or an immediate family member has received research or institutional support and nonincome support (such as equipment or services), commercially derived honoraria, or other non-research–related funding (such as paid travel) from DePuy.

J Am Acad Orthop Surg 2013;21: 440-442

http://dx.doi.org/10.5435/JAAOS-21-07-440

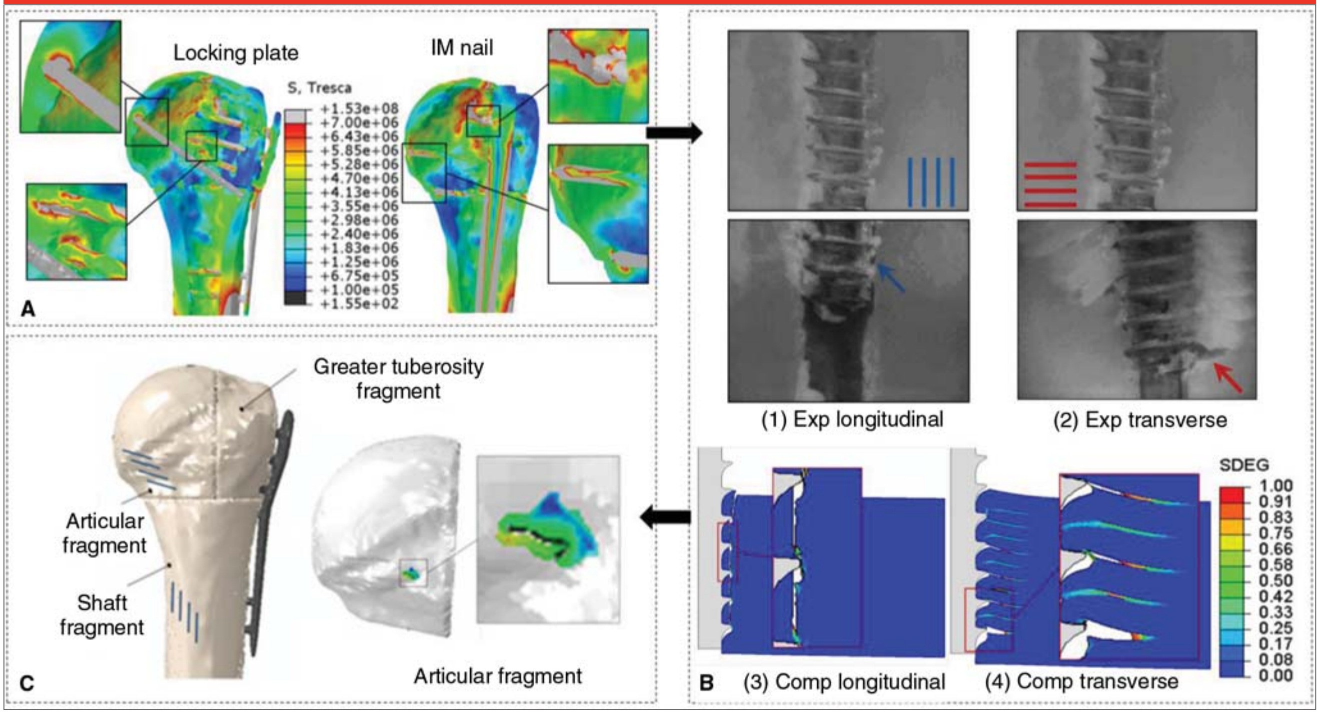
Orthopaedic fracture fixation encompasses a broad range of fracture types. An orthopaedic surgeon may select from a wide array of commercially available devices for the surgical repair of a given fracture type. Complexity of fracture repair increases with the number of fracture fragments that result from the injury and with the level of displacement of these fragments. In the case of proximal humeral fractures, for example, several types of fixation devices have been developed and implemented clinically, ranging from locking plates to intramedullary nails.

Standard finite element analysis of fracture fixation devices models cortical bone as an isotropic elastic material, with material inhomogeneity based on Hounsfield units taken from CT scans.[1] This simplified approach to the modeling of cortical bone allows for a preliminary assessment of device performance in terms of identification of stress concentrations in the device and in the bone. A recent study has extended this method to assess micromotion between fracture fragments.[2] Additionally, the use of emerging device materials such as carbon fiber–reinforced polyetheretherketone has been investigated.[2] Device failure commonly occurs because of pullout/pushout of orthopaedic screws from the surrounding cortical bone.[3] Periprosthetic stress concentrations in the region of screw threads have been identified for locking plate and intramedullary nail simulations (Figure 1, A). Therefore, an improved understanding of the mechanism of bone damage leading to screw pullout is required to develop computational models that will guide the development of next-generation devices.

The alignment of the cortical bone microstructure relative to the screw axis is a critical factor in the force required for screw pullout. When osteons are aligned perpendicular to the screw axis (ie, transverse alignment), the experimentally measured screw pullout force is approximately two times higher than that recorded when osteons are aligned parallel to the screw axis (ie, longitudinal alignment).[4] Furthermore, a novel experimental screw-pullout damage visualization technique has revealed an extensive zone of material damage and crack growth perpendicular to the screw axis for transverse osteon alignment (Figure 1, B, panel 2). In contrast, localized cracks parallel to the screw axis allow for pullout with minimal material damage for longitudinal osteon alignment (Figure 1, B, panel 1). The strong influence of osteon alignment on crack propagation in bone has also been observed in single mode and mixed-mode fracture mechanics tests.[5,6]

To provide reliable analysis and design of next-generation orthopaedic devices, the complex damage and fracture mechanics of bone must be incorporated into finite element models. Feerick and McGarry[4] have demonstrated that influence of osteon alignment on bone fracture during screw pullout can be accurately simulated using a

Figure 1



A, Computational analysis of a proximal humeral fracture repaired with locking plate and intramedullary (IM) nail, highlighting shear stress concentrations in the region of screw tips. The scale is given in MPa, with maximal stresses shown in grey and red.[2] **B,** Panels 1 and 2: Experimental (Exp) study of longitudinal (blue lines) and transverse (red lines) screw pullout using the screw pullout damage visualization technique (arrows).[4] Panels 3 and 4: Computational (Comp) replication of longitudinal and transverse screw pullout. Once the scalar stiffness degradation (SDEG) value reaches 1, the element is removed from the mesh. **C,** Application of extended finite element method to locking plate fixation of proximal humeral fracture.[7] (Panel A reproduced with permission from Feerick EM, Kennedy J, Mullett H, Fitzpatrick D, McGarry P: Investigation of metallic and carbon fibre PEEK fracture fixation devices for three-part proximal humeral fractures. *Med Eng Phys* 2013;35[6]:712-722. Panel B [1], lower left, reproduced with permission from Feerick EM, Liu XC, McGarry P: Anisotropic mode-dependent damage of cortical bone using the extended finite element method [XFEM]. *J Mech Behav Biomed Mater* 2013;20:77-89. Panel B [1], top left; [2], lower right; and [3] and [4] reproduced with permission from Feerick EM, McGarry JP: Cortical bone failure mechanisms during screw pullout. *J Biomech* 2012;45[9]:1666-1672.)

composite damage model, which undergoes plastic yielding and damage initiation when a critical plastic strain is reached. Subsequent bone fracture is simulated using an element removal technique, and computed crack patterns correlate closely to experimental observations for both longitudinal and transverse screw pullout.

The explicit representation of the bone microstructure using the aforementioned composite-damage model provides accurate simulation of the experimental interaction of a single screw with the surrounding cortical bone (Figure 1, B, panels 3 and 4). However, such detailed microstructural modeling is too computationally demanding for the simulation of entire fracture fixation constructs under physiologic loading. To produce a bone fracture model that does not require explicit representation of the bone microstructure, an anisotropic damage criterion has been proposed in a study by Feerick et al,[7] with the extended finite element method (XFEM) implemented to predict crack propagation. This model is shown to capture complex patterns of bone fracture under single mode and mixed-mode loading and correctly predicts crack trajectories during transverse and longitudinal screw pullout. Most importantly, given the computational efficiency of the XFEM method, this model can be used to design and analyze full orthopaedic devices, such as a locking plate for proximal humeral fixation (Figure 1, C). Additionally this computational tool offers the future potential to provide presurgical patient-specific guidance with regard to device selection and optimal screw spacing and screw orientation.

References

1. Maldonado ZM, Seebeck J, Heller MO, et al: Straining of the intact and fractured proximal humerus under physiological-like loading. *J Biomech* 2003;36(12):1865-1873.
2. Feerick EM, Kennedy J, Mullett H, Fitzpatrick D, McGarry P: Investigation of metallic and carbon fibre PEEK fracture fixation devices for three-part proximal humeral fractures. *Med Eng Phys* 2013; 35(6):712-722.
3. Owsley KC, Gorczyca JT: Fracture displacement and screw cutout after open reduction and locked plate fixation of proximal humeral fractures [corrected]. *J Bone Joint Surg Am* 2008; 90(2):233-240.
4. Feerick EM, McGarry JP: Cortical bone failure mechanisms during screw pullout. *J Biomech* 2012;45(9):1666-1672.
5. Norman TL, Nivargikar SV, Burr DB: Resistance to crack growth in human cortical bone is greater in shear than in tension. *J Biomech* 1996;29(8):1023-1031.
6. Zimmermann EA, Launey ME, Barth HD, Ritchie RO: Mixed-mode fracture of human cortical bone. *Biomaterials* 2009;30(29):5877-5884.
7. Feerick EM, Liu XC, McGarry P: Anisotropic mode-dependent damage of cortical bone using the extended finite element method (XFEM). *J Mech Behav Biomed Mater* 2013;20:77-89.