



Standard Test Method for Finite Element Analysis (FEA) of Metallic Orthopaedic Total Knee Femoral Components under Closing Conditions¹

This standard is issued under the fixed designation F3161; the number immediately following the designation indicates the year of original adoption or, in the case of revision, the year of last revision. A number in parentheses indicates the year of last reapproval. A superscript epsilon (ϵ) indicates an editorial change since the last revision or reapproval.

1. Scope

1.1 This standard establishes requirements and considerations for the numerical simulation of metallic orthopaedic cemented and cementless total knee femoral components using Finite Element Analysis (FEA) techniques for the estimation of stresses and strains. This standard is only applicable to stresses below the yield strength, as provided in the material certification.

1.2 *Purpose*—This test method establishes requirements and considerations for the development of finite element models to be used in the evaluation of metallic orthopaedic total knee femoral component designs for the purpose of prediction of the static implant stresses and strains. This procedure can be used for worst-case assessment within a family of implant sizes to provide efficiencies in the amount of physical testing to be conducted. Recommended procedures for performing model checks and verification are provided to help determine if the analysis follows recommended guidelines. Finally, the recommended content of an engineering report covering the mechanical simulation is presented.

1.3 *Limits*—This document is limited in discussion to the static structural analysis of metallic orthopaedic total knee femoral components (which excludes the prediction of fatigue strength).

1.4 The values stated in SI units are to be regarded as standard. No other units of measurement are included in this standard.

1.5 *This standard does not purport to address all of the safety concerns, if any, associated with its use. It is the responsibility of the user of this standard to establish appropriate safety and health practices and determine the applicability of regulatory limitations prior to use.*

2. Significance and Use

2.1 This standard is applicable to the calculation of stresses seen on a knee femoral component when loaded in a manner

¹ This test method is under the jurisdiction of ASTM Committee F04 on Medical and Surgical Materials and Devices and is the direct responsibility of Subcommittee F04.22 on Arthroplasty.

Current edition approved Feb. 1, 2016. Published March 2016. DOI: 10.1520/F3161-16

described in this test method. This method can be used to establish the worst-case size for a particular implant family. When stresses calculated using this method were compared to the stresses measured from physical strain gauging techniques performed at one laboratory, the results correlated to within 9%.

3. Geometric Data

3.1 Finite element models are based on a geometric representation of the device being studied. The source of the geometric details can be obtained from drawings, solid models, preliminary sketches, or any other source consistent with defining the model geometry. In building the finite element model, certain geometric details may be omitted from the orthopaedic implant geometry shown in the Computer Aided Design (CAD) model if it is determined that they are not relevant to the intended analysis. Engineering judgment shall be exercised to establish the extent of model simplification and shall be justified.

3.2 It is most appropriate to consider the worst-case stress condition for the orthopaedic implant family being simulated. The worst-case shall be determined from all relevant engineering considerations, such as femoral component geometry and dimensions. If finite element analysis is being used for determining the worst-case, then the worst-case size may not be known. It may be necessary to run several sizes in order to determine the worst-case. If the FEA results do not conclusively determine the worst-case configuration, a rationale should be included (e.g., additional analysis or physical testing) to justify the worst-case size.

4. Material Properties

4.1 The required material properties for input into an FEA model for the calculation of strains and displacement are modulus of elasticity (E) and Poisson's ratio (ν). These values can typically be obtained from material certification data. It should be noted that the fatigue test is run under load control; the FEA should also be run under load control. When the FEA is run under load control, the modulus of elasticity will not affect the stress calculations under small displacement theory but will affect displacement and strain. The influence of Poisson's ratio on the stress calculations is negligible.

4.2 Ensure that material property units are consistent with geometric units in the CAD model. SI units are the preferred units of measurement.

5. Loading

5.1 The loading location and orientation of the knee femoral component shall be guided by the loading location and boundary conditions described below. The areas of particular interest are the stresses at the posterior aspect of the condyle, anterior notch, and other design-specific critical regions.

5.2 The worst-case condyle shall be loaded. If the weaker condyle cannot be justified, each condyle shall be analyzed individually. Centrally locate a 7.62 mm diameter projected circle over the apex of the posterior articulating surface with the knee femoral component positioned in 90 degrees of flexion. Apply an anterior directed 1 N load uniformly over the face generated by the intersection of this projected circle with the articulating surface. Refer to Fig. 1 and Fig. 2.

NOTE 1—Do not introduce additional solid material to the femoral component model.

NOTE 2—It is recognized that the loading conditions in this test method will not be identical to those of a physical testing standard currently under development. However, the differences in loading conditions (e.g., load application differences; potting level differences; use of bone cement which is not modeled in FEA) do not significantly affect identification of the worst-case stress condition and construct for subsequent bench testing, which is the primary objective of this test method.

5.3 Ensure that load units are consistent with material property units.

6. Boundary Conditions

6.1 The prescribed boundary condition idealizes embedding the anterior flange within a potting medium. The femoral component shall be fixed in all translations on all “embedded” anterior flange surfaces. Refer to Fig. 1 and Fig. 3. A horizontal plane shall be constructed to define a closed perimeter around the anterior flange periphery. Note that the horizontal plane may not be parallel to the anterior flange bone cut face. The use of other stress evaluation levels and/or constraint levels shall be justified.

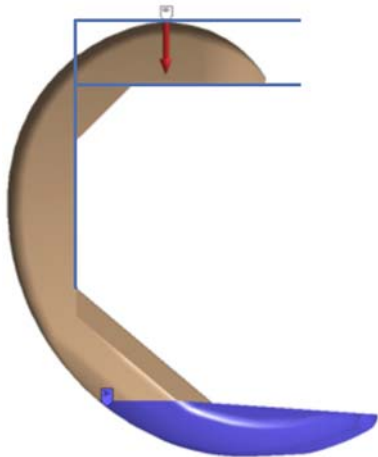


FIG. 1 Apex Location and Anterior Flange Constraint (lateral view)

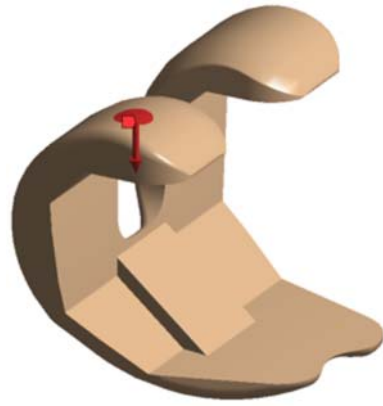


FIG. 2 Apply 1 N Load onto Load Footprint

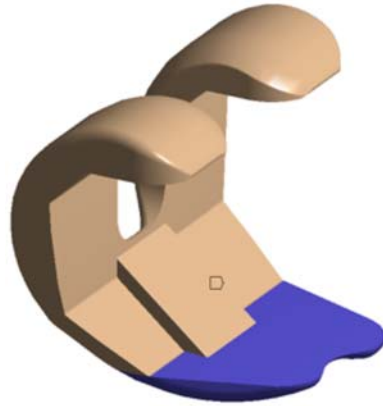


FIG. 3 Anterior Flange Constraint

7. Analysis

7.1 The analysis and modeling system, programs or software used for the finite element model creation and analysis should be capable of fully developing the geometric features and idealizing the loading and boundary condition environment of the orthopaedic implant. An engineering justification shall be provided to support any assumptions and/or simplifications.

7.2 The finite element mesh can be created using automatic meshing, manual meshing, or a combination of the two techniques. The overriding consideration is that the type, the size, and the shape of the elements used must be able to represent the expected behavior without significant numerical limitation or complication. Most FEA packages have a built-in program which checks the shape of the element for the type of analysis selected. If this tool is not available, then additional checks are needed.

7.3 The number and spacing of nodes (i.e. mesh density) should be consistent with the type of element used and the type of result desired. This may be demonstrated with a mesh density study, whereby a series of models with increasing mesh refinement in the critical stress regions is used to demonstrate solution convergence. This allows the error associated with subsequent models to be estimated. The method used to demonstrate mesh convergence, in analysis cases where it is not performed directly onto the model being analyzed, shall be documented in the FEA report. It is recommended that a

minimum of three mesh refinement levels be evaluated and a model convergence of $\leq 5\%$ be demonstrated on all measures and regions of interest. If differences in peak stresses between two sizes in a product family are calculated to be less than 5%, a tightening of the model convergence is recommended to increase the likelihood of establishing the worst-case size within a product family. Reporting of the degrees of freedom is not necessary if the model satisfies the convergence criterion.

7.4 The choice of element type is left to the analyst; however, it is recommended for analysis of a knee femoral component that tetrahedral or hexahedral elements be used. If tetrahedral elements are considered, use of 4-noded elements should be avoided to prevent stress and strain incompatibilities across elements. Additionally, the linear, 4-noded tetrahedron element is a constant strain element. This means that displacement interpolation is linear and the corresponding stresses and strains are constant within any element. Therefore, a very refined mesh is required around locations where high stress/strain gradients are present when utilizing these elements. When elements which are not directly identified in this test method are used, documentation which demonstrates their validity shall be provided in the FEA report.

7.5 The finite element results should be examined to ensure that the geometrical models of the implant, boundary conditions and applied loads have been appropriately defined in the analysis and properly represent the behavior being analyzed. Examples of model behavior which should be examined include the reaction forces and moments as well as the overall deflected shape and deflection magnitude.

7.6 The measure of interest is the Maximum (1st) Principal Stress. Refer to Fig. 4. Stress concentrations near the boundary condition regions are considered to be artifacts and shall not be considered to be regions of interest. If other stress values are used, their validity for use should be documented.

8. Report

8.1 The finite element analysis for the evaluation of an orthopaedic implant should be fully documented in an engineering report. The actual format of the report should comply

with any acceptable proprietary or non-proprietary engineering report format; however, the report shall include, at a minimum, the following:

8.1.1 A complete description of device being analyzed including detailed dimensions. The report should reference a source CAD geometry file by name and revision number. If the evaluation is not being performed on the final design of the device or if there are other significant assumptions that may limit the use of the results, this shall be clearly stated.

8.1.2 A description of boundary constraints, loads, and material properties. The source of the material property data utilized should be referenced.

8.1.3 A summary of the finite element modeling and analysis system used for the analysis. If current versions of widely used, commercially available software are used, this summary can be by name and reference to the version used. For non-commercially available proprietary tools, or custom user modification of commercially available software, sufficient technical background and results of test problems should be provided to demonstrate the utility, verification, applicability and limitations of the software tool.

8.1.4 A description of the procedure used to convert the geometric or CAD representation of the device to the finite element model. Any geometry simplifications should be documented.

8.1.5 A description of the finite element model and its relation to the device being evaluated. The number of nodes and elements (or the degrees of freedom in the model), the finite element type selected including its capabilities, and any special considerations involved in the model should be included. For each region of interest, the maximum (1st) principal stress and von Mises stress at the location of maximum (1st) principal stress shall be reported. Additional stress components can be included and their incorporation shall be justified.

8.1.6 A description of mesh convergence considerations and how they were applied to the analysis.

8.1.7 A description of any numerical considerations or convergence criterion associated with the analysis.

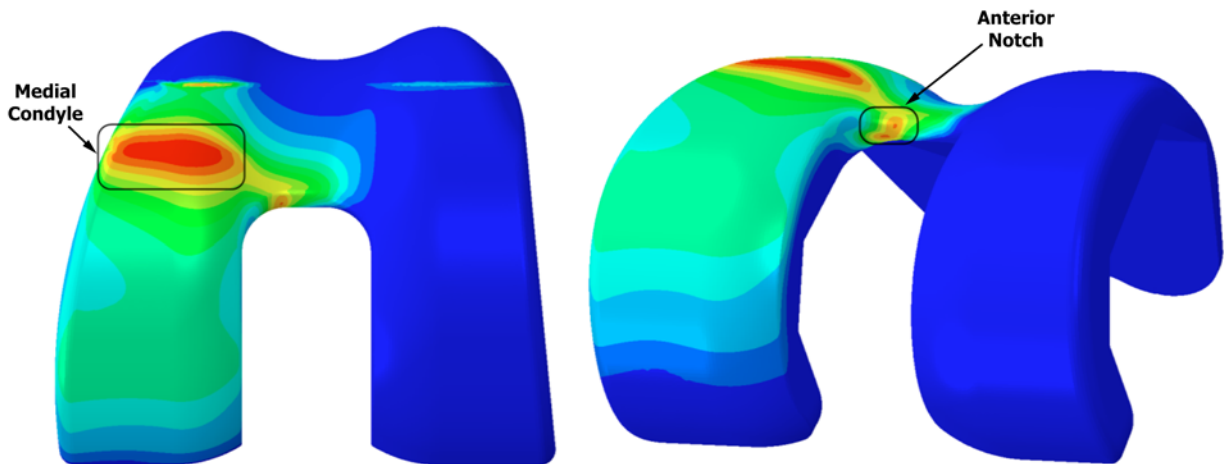


FIG. 4 Stress Plot (arrows point to regions of interest)

8.1.8 A summary of analysis results using all appropriate forms of text, graphics and tabular representations of data to highlight the key behavioral characteristics involved in the evaluation.

8.1.9 Engineering conclusions or recommendations, as appropriate.

8.1.10 Deviations from this standard.

8.1.11 All relevant references and supporting documentation and drawings.

9. Precision and Bias

9.1 The precision and bias of this test method has not been established.

10. Keywords

10.1 computational simulation; displacement; FEA; finite element analysis; model calibration; model validation; model verification; orthopaedic implants; solution sensitivity; strain; stress

APPENDIXES

(Nonmandatory Information)

X1. FEA ROUND ROBIN STUDY

X1.1 A round robin study was performed with 9 labs on a representative knee femoral component model (refer to Fig. 1 and Fig. 2 for geometry) following the procedure in this test method. The stresses in the posterior aspect of the medial condyle and anterior notch were evaluated (Fig. 4). The maximum percent difference from the overall average was less than 1.0% (Table X1.1).

TABLE X1.1 Round Robin FEA Model Results—Maximum Principal Stress (MPa)^{A,B,C}

Round Robin Participant Identifier	Condyle Region Max Principal Stress (MPa)	Notch Region Max Principal Stress (MPa)	% Condyle Stress From Average	% Notch Stress from Average
1A	0.1187	0.1026	-0.01	0.33
1B	0.1188	0.1019	0.07	-0.42
2B	0.1178	0.1018	-0.75	-0.45
3A	0.1189	0.1027	0.16	0.39
4A	0.1188	0.1023	0.03	0.03
4B	0.1190	0.1027	0.20	0.40
5A	0.1188	0.1026	0.10	0.32
6C	0.1189	0.1026	0.16	0.32
7D	0.1190	0.1030	0.24	0.68
8A	0.1182	0.1016	-0.41	-0.69
8B	0.1186	0.1018	-0.09	-0.49
9A	0.1189	0.1020	0.16	-0.30
9B	0.1189	0.1022	0.16	-0.10
Average	0.1187	0.1023		
Standard Deviation	0.0003	0.0004		
Range			-0.75 to 0.24	-0.69 to 0.68

^A All laboratories used 10-noded tetrahedral elements.

^B All laboratories used the recommended convergence criterion of $\leq 2\%$. However, also note that the 2% convergence criterion was not necessarily performed in both regions of interest individually in the round robin. It is recommended that model convergence within each region of interest is $\leq 2\%$ at each location of interest.

^C The designations A-D under “Round Robin Participant Identifier” refer to the 4 FEA software solutions included.

X1.2 The CAD model that was analyzed during the round robin is available for download at <http://www.astm.org/COMMITTEE/F04.htm>. Given this CAD model, an analyst can develop a finite element model consistent with that used in the ASTM Round Robin. Loading and boundary condition

application, as well as a mesh convergence study, can then be performed utilizing the method outlined in this test method. The expectation is that the user will obtain results that are consistent with those reported in **Table X1.1**.

X2. A COMPARISON OF FEA-BASED STRAIN RESULTS TO CONVENTIONAL STRAIN GAGE MEASUREMENTS

X2.1 Introduction:

X2.1.1 The purpose of this experiment was to compare the stresses predicted using finite element analysis (FEA) to those measured using conventional strain gages on actual physical test specimens to validate the ASTM FEA methodology.

X2.1.2 At laboratory 1, knee femoral components representing two different materials were selected to determine if there were any variables that would affect the comparison.

X2.1.3 Both strain gage and finite element analyses were performed under similar loading conditions to compare the stress values generated by each method (**Fig. X2.1**).

X2.2 Description of Samples, Description of Equipment/Apparatus, and Test Method:

X2.2.1 Conventional Strain Gage Technique:

X2.2.1.1 *Laboratory 1*—The following materials were used during testing:

- 1 – cemented CoCrMo commercially available knee femoral component
- 1 – cemented Zr-2.5Nb commercially available knee femoral component
- 4 – strain gages (Vishay Micro-measurements, WK-06-060WR-350)
- 1 – strain gage recorder (Vishay Micro-measurements, model P3)

X2.2.1.2 Knee femoral components were tested and simulated using loading and boundary conditions in this test method. For each component, rosette strain gages were placed in two locations. One gage was placed in the area of the posterior aspect of the condyle and the second was on the anterior notch (**Fig. X2.2**).

X2.2.1.3 The components were potted in bone cement according to the loading configuration to simulate 90° of flexion. The three component strain measurements were recorded for each rosette gage using a 4-channel strain gage recorder in 11.2 N increments up to the maximum load. The results were plotted and curve fitted to determine the stress vs. load relationship. The maximum principal strain was calculated for comparison to the FEA according to **Eq X2.1**:

$$\epsilon_{p,q} = \frac{\epsilon_1 + \epsilon_3}{2} \pm \frac{1}{\sqrt{2}} \sqrt{(\epsilon_1 - \epsilon_2)^2 + (\epsilon_2 - \epsilon_3)^2} \quad (\text{X2.1})$$

X2.2.1.4 Using standard reference values for the Young's modulus (E) of each material, the maximum principal stress (σ) was calculated according to **Eq X2.2** for comparison to the FEA:

$$\sigma = E\epsilon \quad (\text{X2.2})$$

X2.2.1.5 Three-dimensional FEA models of all the components were developed in a commercially available software package. The components were meshed using 10-noded modified tetrahedral elements utilizing a commercially available software package. A load of 2,224 N was applied perpendicular to the posterior aspect of the condyle. The femoral component was fixed in all translations on all “embedded” anterior flange surfaces.

X2.2.1.6 The strain gage measurements and FEA predictions were performed independently by two engineers in the single laboratory and the results were subsequently compared.

X2.3 Participating Laboratory:

X2.3.1 The following laboratory participated in this study:

Smith & Nephew, Inc.
1450 Brooks Road
Memphis, TN 38116

Senior Research Engineer
Patrick Aldinger
901-399-5495
patrick.aldinger@smith-nephew.com

X2.4 Statistical Data Summary:

X2.4.1 Conventional Strain Gage Technique:

X2.4.1.1 A summary of the results of the strain gage and finite element analyses are shown in **Table X2.1**. Representative pictures of the strain gage setup and FEA results are shown in **Fig. X2.2** (from Laboratory 1).

X2.4.1.2 While the strain gage analysis provides a single value for stress at the location of the gage, in reality it is a sum over an area formed by the gage length of the strain gage. For the strain gages used in this experiment, the area is approximately 2.32 mm². In order to best approximate the area effect on the FEA results, the values of the nodes over a similar area were averaged to determine a mean stress value.

X2.4.1.3 At laboratory 1, the strain gage values and FEA predicted stresses were within 4.1%, on average. The maximum error observed with any of the strain gages was 9.4%, and the minimum error observed was 1.3%.

TABLE X2.1 Laboratory 1—Percentage Difference between Strain Gage Measured and FEA Calculated Stresses on Two Different Knee Femoral Components

Finish	Material	Location	% Difference
Polished	CoCrMo	posterior condylar bend	1.3%
		anterior notch	3.7%
Polished	Zr-2.5Nb	posterior condylar bend	2.1%
		anterior notch	9.4%
		Average	4.1%
		Range	1.3% - 9.4%

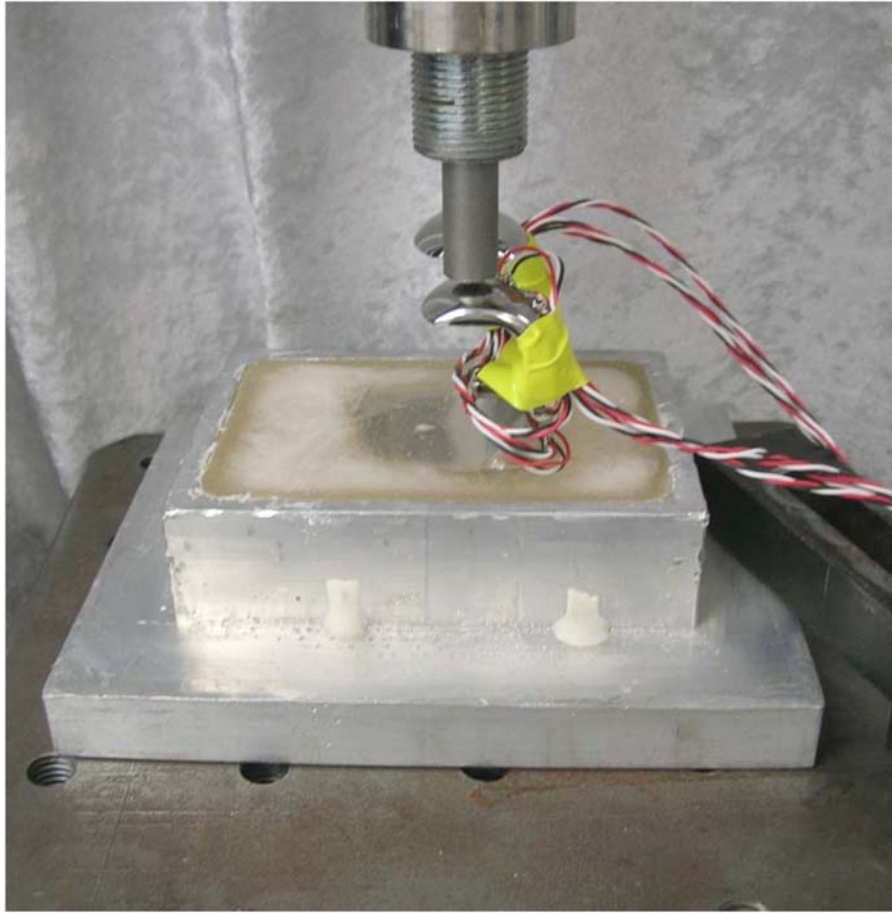
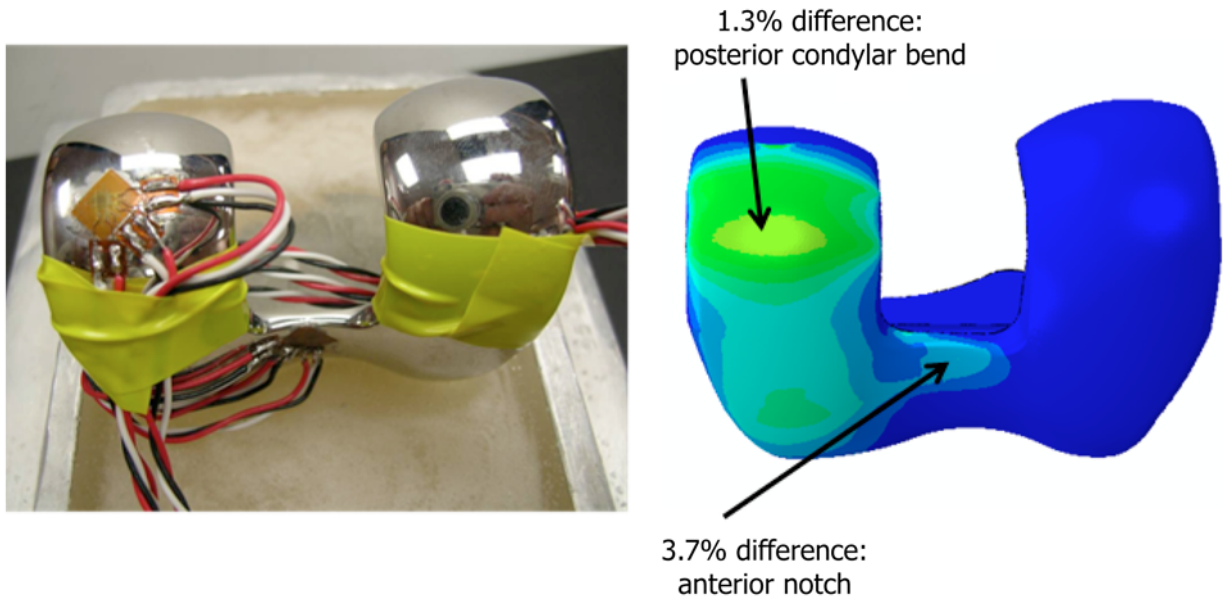


FIG. X2.1 Laboratory 1—Knee Femoral Component Test Set-up



NOTE 1—This figure illustrates a test performed solely to correlate FEA to physical test results.

FIG. X2.2 Laboratory 1—Conventional Strain Gage (left) and FEA (right) Comparison for the Polished CoCr Femoral Component

ASTM International takes no position respecting the validity of any patent rights asserted in connection with any item mentioned in this standard. Users of this standard are expressly advised that determination of the validity of any such patent rights, and the risk of infringement of such rights, are entirely their own responsibility.

This standard is subject to revision at any time by the responsible technical committee and must be reviewed every five years and if not revised, either reapproved or withdrawn. Your comments are invited either for revision of this standard or for additional standards and should be addressed to ASTM International Headquarters. Your comments will receive careful consideration at a meeting of the responsible technical committee, which you may attend. If you feel that your comments have not received a fair hearing you should make your views known to the ASTM Committee on Standards, at the address shown below.

This standard is copyrighted by ASTM International, 100 Barr Harbor Drive, PO Box C700, West Conshohocken, PA 19428-2959, United States. Individual reprints (single or multiple copies) of this standard may be obtained by contacting ASTM at the above address or at 610-832-9585 (phone), 610-832-9555 (fax), or service@astm.org (e-mail); or through the ASTM website (www.astm.org). Permission rights to photocopy the standard may also be secured from the Copyright Clearance Center, 222 Rosewood Drive, Danvers, MA 01923, Tel: (978) 646-2600; <http://www.copyright.com/>