

NASA/CR—2002-211277



Procedure for Tooth Contact Analysis of a Face Gear Meshing With a Spur Gear Using Finite Element Analysis

George Bibel
University of North Dakota, Grand Forks, North Dakota

January 2002

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA Access Help Desk at 301-621-0134
- Telephone the NASA Access Help Desk at 301-621-0390
- Write to:
NASA Access Help Desk
NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

NASA/CR—2002-211277



Procedure for Tooth Contact Analysis of a Face Gear Meshing With a Spur Gear Using Finite Element Analysis

George Bibel
University of North Dakota, Grand Forks, North Dakota

Prepared under Cooperative Agreement NCC3-904

National Aeronautics and
Space Administration

Glenn Research Center

January 2002

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100

Available electronically at <http://gltrs.grc.nasa.gov/GLTRS>

PROCEDURE FOR TOOTH CONTACT ANALYSIS OF A FACE GEAR MESHING WITH A SPUR GEAR USING FINITE ELEMENT ANALYSIS

George Bibel
Department of Mechanical Engineering
University of North Dakota
Grand Forks, North Dakota 50202

SUMMARY

A procedure was developed to perform tooth contact analysis between a face gear meshing with a spur pinion using finite element analysis. The face gear surface points from a previous analysis were used to create a connected tooth solid model without gaps or overlaps. The face gear surface points were used to create a five tooth face gear Patran model (with rim) using Patran PCL commands. These commands were saved in a series of session files suitable for Patran input. A four tooth spur gear that meshes with the face gear was designed and constructed with Patran PCL commands. These commands were also saved in a session files suitable for Patran input. The orientation of the spur gear required for meshing with the face gear was determined. The required rotations and translations are described and built into the session file for the spur gear. The Abaqus commands for three-dimensional meshing were determined and verified for a simplified model containing one spur tooth and one face gear tooth. The boundary conditions, loads, and weak spring constraints were determined to make the simplified model work. The load steps and load increments to establish contact and obtain a realistic load was determined for the simplified two tooth model. Contact patterns give some insight into required mesh density. Building the two gears in two different local coordinate systems and rotating the local coordinate systems was verified as an easy way to roll the gearset through mesh. Due to limitation of swap space, disk space and time constraints of the summer period, the larger model was not completed.

INTRODUCTION

Face gears for use in helicopter transmissions were studied. The face gear has several advantages over the traditional spiral bevel gear. Two advantages are reduced sensitivity to misalignment and reduced noise from low transmission error (ref. 1). The face gear tooth surface geometry is based on the kinematics of the generating action of the pinion shaper. The instantaneous line of contact on the pinion tooth is defined by the development of the equation of meshing. The face gear tooth surface is derived by coordinate transformation of its contraform on the pinion surface (refs. 1 and 2).

The existing design methods for face gear tooth stresses are simple modifications of spur gear stress analysis programs. It is difficult to validate these design procedures with strain gages because face gear teeth are very small. Therefore it is desirable to develop finite element techniques to validate the existing face gear design technology. The work reported here describes a process for doing three-dimensional contact analysis using the finite element method of a face gear contacting a mating spur gear. Described are the problems associated with geometry, meshing and convergence of the contact analysis.

ANALYSIS PROCEDURE

Face Gear Surface Points

The description of the face gear surface, as given by the analysis from reference 1, consists of 30 points on ten different sections (corresponding to ten different Z values).

The input to this analysis was as follows:

```
Input TN1P, TN2 and TN1, where
    TN1P is the pinion number of teeth,
    TN2 is the gear number of teeth,
    TN1 is the number of teeth of the shaper:
17,69,18
Input diametral pitch:
10
Input the shaft angle (degrees):
90
Input the pressure angle (degrees):
27.5
Input: e(mm), g(minutes), and q(mm), where
    e is the misalignment in the direction mutually perpendicular to the plane formed by the
        axes of the gear and pinion,
    g is the error of the shaft angle,
    q is the misalignment along the gear axis:
0.001,0.001,0.001
```

Points 1 to 20 define the tooth face. Points 21 to 30 define the tooth fillet. Points 20 and 21 coincide. A surface made of all points from the analysis is seen in figure 1. Figure 2 shows the points on the inner most section and the points on the outer most section. In figures 1 and 2, it is seen that some points on the fillet wrap around onto the fillet of the adjacent tooth.

The axis of rotation of the face gear as defined by the analysis is the Y axis (see fig. 3). The first point of each tooth sections always starts at $Y = 0$. All subsequent points have increasing negative Y values. The largest negative Y value corresponds to the bottom of the fillet. When the negative Y value starts to increase from the bottom of the fillet, the points are wrapping around onto the fillet of the next tooth. The points on the fillet of the adjacent tooth are discarded from the model. The points used on each section are 1 to 20 to define the tooth face and points 22 through the largest negative Y value to define the fillet. The number of points used to identify the fillet varies from six to zero.

The first 20 points on each of ten sections (200 points total) was used to define the tooth face. Points 21 to 30 on each section are not used initially. These 200 points were input with Patran PCL in the session file `LTooth.ses`. The session file `LTooth2.ses` creates curves through the 200 points on the tooth face.

Points 22 to 30 on each section are used to identify the fillet. Not all of the eight points are used to identify the fillet. Only those points up to the highest negative Y value. The points used to identify the fillet are given in the Patran session file `LFillet.ses`. This file also creates curves through the points on the fillet.

`LSurface.ses` makes all of the surfaces on the tooth face and fillet. `Solids.ses` mirrors all of the surfaces about the minus YZ plane (0.1 plane in Patran notation) and makes the solids for the tooth and fillet.

When this tooth is rotated to create an adjacent tooth, the points do not line up in the fillet region. This can be understood by looking at the spacing of the points in the fillet region in figure 2. The spacing is uniform and not designed to identify the true bottom of the fillet. The actual spacing for the first section fillet points is about 0.010 in. The “true” bottom of the fillet is not accurately defined. The maximum gap and overlap is about 0.0035 in. or about a third of the spacing used. By adjusting the points on the bottom of the fillet to lie on the same radial line ($\theta = (0.5)360/(69 \text{ teeth})$), the gaps and overlaps can be eliminated. Two points were adjusted about 0.003 in. and two points were adjusted 0.002 in. The remaining six points were adjusted 0.001 in or less.

The following sessions files are played in the order listed:

`gearA.ses`, `gearB1.ses`, `gearB2.ses`, `gearC1.ses`, `gearC2.ses`, and `gearC3.ses`. All of these files have been combined into `FaceGear.ses`.

The face gear consists of the first tooth plus two teeth rotated clockwise and two teeth rotated counterclockwise for a total of five teeth.

Spur Gear Geometry

The following data was used as input to the NASA Glenn computer program `Gpat2a.exe`, which was used to define the spur gear geometry (ref. 3):

17	Number of teeth on gear
N	N=External gear
1000	Number of teeth on cutter
27.5	Pitch line pressure angle, deg
10	Diametral pitch, teeth/in
Y	Y=Standard teeth
0.65	Radius for 0.2 backup ratio, in
0.5475	Slot outer radius, in
0.35	Hub outer radius, in
0.26	Hub inner radius, in

The spur gear consists of the first tooth plus one tooth rotated clockwise and two teeth rotated counterclockwise for a total of four teeth. The file used to create the spur gear and rotate and translate into mesh are given in the Patran session file `Spur.ses`. Running `Spur.ses` after `FaceGear.ses` will generate the complete model. Because of limitation of time, disk space, and swap file errors, the “big” model was not used. Verification of three-dimensional contact with Abaqus was done on a simpler model consisting of one face gear tooth and one spur gear tooth. The files used to create this smaller model are `FaceS.ses` and `sSpur.ses`.

When generating the spur gear, answer yes to all questioned asked by Patran.

Orientation of the Two Gears in Mesh

Figure 4 shows the orientation of the face gear and spur gear. The face gear is centered at the global origin with the Y axis being the axis of rotation. The two-dimensional profile of the spur gear, as generated by the program, lies in the YX plane. The Z axis is the axis of rotation of the spur gear. As shown in figure 4, the spur gear profile is extruded 1.285 in. in the Z direction to establish the depth of the spur gear. The center of the face-gear tooth face is at $Z = 3.501427$ in., the spur gear must be translated $Z = +2.858927$ in. to center on the face gear ($2.858927 = 3.501427 - 1.285/2$).

Since the face gear is in the ZX plane, and the spur gear profile is in the XY plane, with equal parts above and below the X axis, the spur gear must be translated in $+Y$ direction to be in mesh. This translation equals the spur gear radius minus the face gear tooth height plus a clearance of 0.030 in. or $Y = +0.7552$ in. ($Y = 0.95 - 0.2248 + 0.03$).

In addition to the above translations, the tooth must be rotated into mesh. The above translations result in a tooth on top of the spur gear. A tooth on the bottom of the spur gear is needed for mesh. Rotating a spur gear tooth 180° results in a tooth that is one half tooth out of sequence for meshing. One half tooth rotation corresponds to one half of $360/(17 \text{ teeth}) = 10.588235^\circ$. Therefore, the total rotation to obtain mesh is 190.588235° .

The actual values of the Y translation and the Z axis rotation required for mesh are somewhat arbitrary. Final mesh is verified by viewing the mesh in Patran (see fig. 5). Zooming on figure 5 verifies no interference between the spur and face gear.

A summary of the translations and rotations to obtain the spur gear and face in mesh are as follows:

1. Translate the spur gear in the $+Z$ direction 2.858927 in. to center on face gear tooth.
2. Translate the spur gear in the $+Y$ direction 0.7552 in.
3. Rotate the spur gear 190.588235° to engage it in mesh.

The translations for the spur gear were obtained by defining an appropriate local coordinate system. Local coordinate systems were also defined as an easier way to rotate the gear through mesh and will be discussed later.

Boundary Conditions

Since the face gear is centered at the reference global system, it is easier to fix the spur gear and constrain the face gear with a rigid link to rotate about its axis of rotation, the Y axis. This was done with rigid beams between the inner diameter of the face gear and the axis of rotation for the face gear. The load was applied as point loads to the face gear. A weak spring was used to constrain the rotation of the face gear (and oppose the point loads applied to the face gear).

Because the face gear was modeled almost to its inner diameter, the pie sector of the face gear narrows as the inner diameter approaches $Y = 0$. There is concern this narrow sector is flexing. It may be more appropriate to fix the edge surfaces of the face gear and constrain the spur gear to rotate about its axis of rotation.

The simplified two tooth model that successfully ran did not use rigid beams to force rotation of the face gear about its axis of rotation. Two nodes on the inner diameter of the face gear were fixed with zero translation in all directions. These two points defined an axis of rotation for the single tooth face gear model. Those two points are about 0.25 in. from the true axis of rotation.

Using *MPC (multipoint constraint) to make a rigid link may be a better way to constrain one of the gears to rotate about its axis of rotation.

Three-Dimensional Contact With Abaqus

The following steps are required to do three-dimensional contact with Abaqus.

1. Identify the surfaces that contact.

```
*SURFACE DEFINITION, NAME=FACETOOTH  
F1,
```

FACETOOTHONE is a user designated name. F1 is the set of elements on (i.e., ELSET), for example, the face gear tooth 1 contact region.

The ELSET of elements involved in contact must be identified. This was done by applying a minor pressure load within Patran to the tooth face. The Abaqus input deck will then have an ELSET identified and available for surface definition. On the large nine tooth model the pressure loads were designated $f1$ to $f5$ on the five face gear teeth and $s1$ to $s4$ on the four spur gear teeth. When a load is applied on the face gear in the positive X direction, surface $s1$ contacts with $f1$, $s2$ with $f2$, etc.

Surface definition must always occur in pairs. At least one surface must be identified on the face gear and one surface identified on the spur gear. If two pairs of teeth are to contact, then there will be four surface definition commands. The surface definition for the spur tooth might look as follows.

```
*SURFACE DEFINITION, NAME=SPURTOOTH  
S1,
```

2. Identify a contact pair. This command is used to identify pairs of surfaces that interact with each other.

```
*CONTACT PAIR, SMALL SLIDING, INTERACTION=MYANALYSIS  
FACETOOTH, SPURTOOTH
```

MYANALYSIS is a user given name. FACETOOTH and SPURTOOTH are the user given names given in the surface definition commands.

When a contact pair contains two deformable surfaces, the user must choose which surface will be the slave surface and which will be the master. The master surface is the stiffer structure or structure with coarser mesh if the two bodies are of the same stiffness. The name of the slave surface is the first data item on the *CONTACT PAIR command.

```
*CONTACT PAIR, INTERACTION=NAME  
SLAVE_SURFACE_NAME, MASTER_SURFACE_NAME
```

With the spur gear fully constrained, it was designated the master surface.

After an unsuccessful initial try, additional optional parameters were added to the *CONTACT PAIR command. These parameters were HCRIT= .01 and ADJUST=0.001. These parameters did not help the analysis

converge. They were left in the input deck and described here in an attempt to thoroughly describe the input deck that did run.

3. Define surface interaction properties.

```
*SURFACE INTERACTION, NAME=MYANALYSIS
```

Where MYANALYSIS is the name given by the user in the *CONTACT PAIR command.

4. All of the above commands must be entered into the input deck above the *STEP command.
5. The *STEP command created by Patran must be modified to the following:

```
*STEP, NLGEOM, INC=50
```

NLGeo is short for nonlinear geometry. This indicates a large deformation problem. The deformation of contact is large compared to an elastic deformation of steel. INC=50 means the load step is broken up into 50 substeps or increments to help the solution converge.

It is difficult to get a contact problem to converge on the first loadstep. This is because the model is floating and free to accelerate until contact is established. When the problem was not converging the following error and warning messages occurred in Abaqus.

```
***WARNING: OVERCLOSURE OF CONTACT SURFACES FACE and SPUR IS TOO SEVERE --  
CUTBACK WILL RESULT. YOU MAY WANT TO CHANGE THE VALUE OF HCRIT ON  
THE *CONTACT PAIR OPTION.
```

The above warning occurred many times and ultimately resulted in the following error message indicating non-convergence in the ten increments initially used (i.e., INC = 10)

```
***ERROR: TOO MANY INCREMENTS NEEDED TO COMPLETE THE STEP
```

To overcome these error messages, a very small load and a high value for INC was used. The analysis finally ran with a 5 lb load and INC = 50. INC = 50 makes the problem iterate and run longer. A smaller value will speed up the analysis, but the solution may not converge. The 5 lb force was applied in Patran and appeared in the Abaqus input deck.

6. Apply a realistic load with a second load step. Once contact is successfully established, the model is now stiffer and can withstand a realistic load. This was done by copying everything that appeared between the *STEP command and the *END STEP command and pasting it after the existing *END STEP command. Several NSET and *END STEP commands are contained between the *STEP and *END STEP command. These sets of nodes and elements were deleted. Presumably it would not hurt to leave it in. Also the load on the nodes was changed from 1 lb per node to 50 lb.

Summary Of Procedure Required To Duplicate Successful Three-Dimensional Analysis (The Two Tooth Model)

1. Enter Patran and run Faces . ses to make face tooth.
2. Run sSpur . ses to make spur tooth and answer yes to all questions.
3. Create the Abaqus input deck and edit as follows:
 - A. Add the contact commands as explained above.
 - B. Modify the *STEP command as explained above.
 - C. Add a second load step as explained above. The load must be increased as explained above.

The following changes were made to the input deck for the model with one tooth on each gear: (For one load step with 5 lb load on face gear).

```

*SURFACE DEFINITION, NAME=FACE
F1,
*SURFACE DEFINITION, NAME=SPUR
S1,
*CONTACT PAIR, SMALL SLIDING, INTERACTION=AAA, HCRIT=.01, ADJUST=0.001
FACE, SPUR
*SURFACE INTERACTION, NAME=AAA
** STEP 1, DEFAULT STATIC STEP
** LOADCASE, DEFAULT
**
*STEP, NLGEOM, INC=50

```

NOTE: The existing *STEP command must be removed and replaced with the one shown above. HCRIT and ADJUST options were added when it was not converging. These options may not be needed.

The two tooth model input deck is `Pair2.inp`. This model was made from `Face.ses` and `sSpur.ses`. The `*.inp` file was modified by adding the contact commands above and by changing the *STEP command as shown. The first attempt on this model was a 1-lb force applied to five nodes (five lb. total force) with `INC=10` on the *STEP command. This attempt resulted in the warnings and error messages described above.

When `INC` was changed to 50, the problem did run to completion. The resulting maximum stress was 2600 psi with contact at one node in the upper corner of the face gear.

After contact is established, more load can be applied in a second load step. To apply the second load step, everything between *STEP and *END STEP was copied and pasted into the input deck. The `ELSET` and `NSET` definitions were deleted as redundant. The force on the five nodes were increased from 1 to 50 lbs. Contact increased to three nodes. The two load step input deck is `Pair2a.inp`.

RESULTS AND DISCUSSION

Results For Two Tooth Model (Using C3D8, Eight-Node Linear Brick Element)

The first successful run had one load step with 5 lb of force applied. Contact was at one node. The contact stresses were about 2600 psi. Since the actual load should be about 194 times larger and the load should be shared with two pairs of teeth, this stress appears realistic.

The second run had a second load step with 250 lb applied. This is about 1/4 of the design load. Contact spread to 3 nodes. The contact pattern at first appears to skip nodes especially when viewing the stress contours on the spur gear. However when looking at the contact pattern on the face gear it can be seen that the pattern actually cuts across a diagonal of a single row of four elements as illustrated in figure 6. This indicates the mesh is too coarse and the contact pattern is flat.

For the case with 250 lb applied force, the contact stresses increased to 46,000 psi. This is considered realistic for such a coarse model. The contact stress pattern is shown in figure 7 for the spur gear and figure 8 for the face gear. The contact pattern of two nodes contacting across the diagonal of a row of four elements remained. This implies a 4 by 2 increase in mesh density should give adequate results. (i.e., four times the number of elements along the height of the teeth and two times the number of elements along the length of the tooth.) This recommendation is based on the results of FEA modeling and favorable strain gage comparison done on spiral bevel gears (ref. 4). A 4 by 2 increase in mesh density should give about twice as many contacting nodes as in the spiral bevel study. Increasing the mesh density and using the nine tooth model (instead two teeth) will result in models 4 by 2 by $9/2 = 36$ times bigger.

Figures 9 to 11 show the Patran “seeds” used to create the mesh in the face gear and spur gear.

Meshing Action Obtained With Rotation Of Local Coordinate Systems

One method of rotating the gears through mesh is to build each gear in its own local coordinate system. The local coordinate systems are then rotated, as required for meshing, in the Patran session file. This results in the rotation of the two gears in the global coordinate system, the system in which the analysis occurs. This is far easier than rotating the gears in Patran and redoing all contact surface definitions and boundary conditions.

The two local coordinate systems are located at the center of each gear. The local coordinate system for the face gear is the same as the global coordinate system. In Patran this local coordinate system was designated coordinate system 1 (The default global coordinate system is coordinate system 0).

The origin for local coordinate system for the spur gear is at $X = 0$, $Y = 0.7552$ in., $Z = +2.858927$ in. This origin reflects the translations required to put the spur gear in mesh with the face gear described above.

In Patran, a local coordinate system is defined by three points:

1. The origin.
2. A point on axis 3.
3. A point on the 1–3 plane.

Since the local coordinate system for the face gear corresponds to the global coordinate system, the three points that define the local coordinate system for the face gear are: $[0\ 0\ 0]$ $[0\ 0\ 1]$ $[1\ 0\ 0]$.

The axis of rotation of the face gear is the Y axis. For rotation of the face gear about the Y axis, the point on axis 3 and the point on the 1–3 plane must change. To rotate the face gear $+1^\circ$ counterclockwise, the following modifications are made to the definition of local coordinate system 1. The point within Patran used to define axis 3 becomes $X_{new} = -\sin\theta$, $Y_{new} = 0$ (same), $Z_{new} = \cos\theta$. The point in Patran used to define the 1–3 plane becomes , $X_{new} = \cos\theta$, $Y_{new} = 0$ (same), $Z_{new} = \sin\theta$. For a -1° rotation clockwise, the 3 axis is defined by $[0.0174524, 0, 0.99984769]$ and the 1–3 plane is defined by $[0.99984769, 0, -0.0174524]$. A \pm rotation of the face gear could have been done. -1° clockwise rotation was arbitrarily chosen.

The following three points were used to define the local coordinate system for the spur gear. Point $[0, 0.7552, 2.858927]$. Point 2 $[0, 0.7552, 3.858927]$. Point 3 $[1, 0.7552, 2.858927]$. The origin includes the translations required to mesh the spur gear with the face gear as described in the section Orientation of the Two Gears in Mesh. The local coordinate system for the spur gear was arbitrarily designated in Patran as local coordinate system 3.

The spur gear rotates about its local Z axis. To rotate the spur gear, only the point on the 1–3 plane must be redefined. The new point to define the 1–3 plane is: $X_{new} = \cos\theta$, $Y_{new} = Y_{old} + \sin\theta$, $Z_{new} = Z_{old}$.

For every -1° rotation of the face gear (about its local Y axis), the spur gear will rotate $69/17 = +4.058823529^\circ$ about its local Z axis. The minus/plus relationship occurs because of the direction the spur gear Z axis points. The new point to describe the 1–3 plane of the spur gear for a $+4.058823529^\circ$ rotation is: $[0.9974919, 0.8259806, 2.858927]$.

The patran session files `FaceS . ses` and `sSpur . ses` contain the changes to the local coordinate system to obtain a -1° rotation of the face gear and a $+4.058823529^\circ$ rotation of the spur gear. However, in the current version of these files these changes are commented out.

The final verification of rolling the gears through mesh in this manner is to rerun the analysis and verify a change in the contact pattern. A shift in the contact pattern is seen in figure 12 (compare to fig. 8). A second verification is the teeth appear to be rotated, and in mesh, when the local coordinate systems are rotated as described above.

Problems Encountered

The following problems were encountered during the development of the analysis procedure:

1. Every time a load was made, a corresponding load case was also made. This reset the default load case resulting in most of the loads not showing up in the Abaqus input deck. One approach is to create no load cases, then all loads will be in the default load case.
2. Edge selection for seeding stopped working. Plotting of solids only helped some, but edge selection was still very erratic. This was a big problem and difficult to work around. Presumably some setting got reset. No resolution to this problem was ever found.

CONCLUSIONS

A procedure was developed to perform tooth contact analysis between a face gear meshing with a spur pinion using finite element analysis. The face gear surface points from a previous analysis were used to create a five tooth face gear Patran model (with rim) using Patran PCL commands. A four tooth spur gear that meshes with the face gear was designed and constructed with Patran PCL commands. The Abaqus commands for three-dimensional meshing were determined and verified for a simplified model containing one spur tooth and one face gear tooth. The following conclusions were derived:

1. Three-dimensional Contact analysis of face gears using Patran and Abaqus was successfully performed. The modifications of the Abaqus input deck as made by Patran are reviewed in this report.
2. Creating each gear in its own local coordinate system is an easier way to roll the gears through mesh. Meshing can then be accomplished by the appropriate rotations of the two local coordinate systems. The correct rotations of the local coordinate systems are the same as would be applied to the two gears.
3. The contact pattern of the gears studied here indicate a much finer mesh was required.

APPENDIX A - List Of All Computer Files

All Patran session files are in directory: /home/s01gb/ses.

1. FaceGear . ses. Patran session file that makes five tooth face gear. FaceGear . ses is made of the following incremental session files:

- LTooth . ses
- LTooth2 . ses
- LFillet . ses
- LSurface . ses
- Solids . ses
- gearA . ses
- gearB1 . ses
- gearB2 . ses
- gearC1 . ses
- gearC2 . ses
- gearC3 . ses

2. FaceS . ses. Patran session file that makes a one tooth face gear.
3. Spur . ses. Patran session file that makes four tooth spur gear.
4. sSpur . ses. Patran session file that makes one tooth spur gear.

All Abaqus files are in directory: /home/s01gb/abaq.

5. Pair2 . inp. One load step model with 5 lbs total load.
6. Pair2a . inp. Two load step model with 250 lbs total load.
7. ShortPair2aa . inp. Two load step input deck shortened for printing and attached to this report in appendix.

The output files were not saved because they were too big and can easily be remade from the input files. Examples of file sizes are:

Pair2 . dat	113990039 bytes
Pair2 . fil	186186168 bytes (6931 nodes)
Pair2a . dat	146601391 bytes
Pair2a . fil	239111352 bytes (6931 nodes)

APPENDIX B - Abbreviated Abaqus Input Deck
(file ShortPair2aa.inp)

*PREPRINT, ECHO=NO, MODEL=NO, HISTORY=YES, CONTACT=YES
*HEADING, SPARSE

ABAQUS job created on 27-Jul-01 at 12:07:49

**

*NODE

1,	-1.5,	-0.3,	2.1
2,	0.0367137,	0.,	3.16038
3,	0.0407408,	-0.0156476,	3.16038
.			
.			
.			
6929,	0.0367532,	0.158814,	2.85893
6930,	-2.36656E-7,	0.2077,	2.85893
6931,	-1.67855E-7,	0.15645,	2.85893

**

**

*ELEMENT, TYPE=C3D8, ELSET=FACEPROP

1,	597,	598,	603,	602,	607,	608,
613,	612					
2,	598,	599,	604,	603,	608,	609,
.						
.						
.						
2532,	3065,	3066,	3073,	3072,	3079,	3080,
3087,	3086					
2533,	3066,	3067,	3074,	3073,	3080,	3081,
3088,	3087					

*ELEMENT, TYPE=SPRINGA, ELSET=SPRING

2101, 2947, 1

*ELEMENT, TYPE=C3D8, ELSET=SPURPROP

2534,	3399,	3400,	3404,	3403,	3463,	3464,
3468,	3467					
2535,	3398,	3399,	3403,	3402,	3462,	3463,
.						
.						
.						
5292,	6676,	6677,	6184,	6183,	6684,	6685,
6196,	6195					
5293,	6672,	6673,	6677,	6676,	6680,	6681,
6685,	6684					

**

** spurFEA

**

*ELSET, ELSET=SPURFEA, GENERATE

2534, 5293, 1

**

** FaceProps

**

*SOLID SECTION, ELSET=FACEPROP, MATERIAL=FACESTEE

1.,

**

** spring

**

*SPRING, ELSET=SPRING

```

10.,
**
** spurProp
**
*SOLID SECTION, ELSET=SPURPROP, MATERIAL=SPURSTEE
1.,
**
** faceSteel
** Date: 20-Jul-01      Time: 13:26:23
**
*MATERIAL, NAME=FACESTEE
**
*ELASTIC, TYPE=ISO
3.E+7,      0.3
**
** spurSteel
** Date: 20-Jul-01      Time: 14:24:45
**
*MATERIAL, NAME=SPURSTEE
**
*ELASTIC, TYPE=ISO
3.E+7,      0.3
**
** edgeY
**
*BOUNDARY, OP=NEW
EDGEY, 2,,      0.
**
** RotationConst
**
*BOUNDARY, OP=NEW
ROTATION, 1,,      0.
ROTATION, 2,,      0.
ROTATION, 3,,      0.
**
** spring anchor
**
*BOUNDARY, OP=NEW
SPRING_A, 1,,      0.
SPRING_A, 2,,      0.
SPRING_A, 3,,      0.
**
** spurConstraints
**
*BOUNDARY, OP=NEW
SPURCONS, 1,,      0.
SPURCONS, 2,,      0.
SPURCONS, 3,,      0.
**
** SURFACE DEFINITION, NAME=FACE
F1,
**
** SURFACE DEFINITION, NAME= SPUR
S1,
**
** CONTACT PAIR, SMALL SLIDING, INTERACTION=AAA, HCRIT=.01, ADJUST=0.001
FACE, SPUR
**
** SURFACE INTERACTION, NAME=AAA
** Step 1, Default Static Step
** LoadCase, Default

```

```

**
*STEP, NLGEOM, INC=50
**
**
This load case is the default load case that always appears
**
*STATIC
**
*NSET, NSET=EDGEY, GENERATE
 1637, 1637, 1
 1954, 1954, 1
 1957, 1957, 1
 1960, 1960, 1
 1963, 1963, 1
 1966, 1966, 1
 2529, 2552, 1
 2697, 2702, 1
 2739, 2744, 1
 2781, 2786, 1
 2823, 2828, 1
 2865, 2870, 1
 2907, 2912, 1
 2949, 2949, 1
 2956, 2956, 1
 2963, 2963, 1
 2970, 2970, 1
 2977, 2977, 1
 2984, 2984, 1
 2991, 2991, 1
*NSET, NSET=ROTATION
 3075, 3334
*NSET, NSET=SPRING_A
 1,
*NSET, NSET=SPURCONS
 6136, 6139, 6142, 6145, 6148, 6151, 6154, 6157,
 6160, 6163, 6166, 6169, 6172, 6175, 6178, 6181,
 6184, 6187, 6190, 6193, 6196, 6199, 6202, 6205,
 6208, 6211, 6214, 6217, 6220, 6223, 6226, 6229,
 .
 .
 .
 6801, 6805, 6810, 6811, 6816, 6817, 6822, 6823,
 6828, 6829, 6834, 6835, 6840, 6841, 6846, 6847,
 6852, 6853, 6858, 6859, 6864, 6865, 6870, 6871,
 6876, 6877, 6882, 6883, 6888, 6889, 6894, 6895,
 6900, 6901, 6906, 6907, 6912, 6913, 6918, 6919,
 6924, 6925, 6930, 6931
*NSET, NSET=FORCEX, GENERATE
 2685, 2689, 1
*ELSET, ELSET=F1
 1, 2, 5, 10, 17, 26, 37, 43,
 109, 110, 113, 118, 125, 134, 145, 151,
 217, 218, 221, 226, 233, 242, 253, 259,
 325, 326, 329, 334, 341, 350, 361, 367,
 .
 .
 .

```

```

1513, 1514, 1517, 1522, 1529, 1538, 1549, 1555,
1621, 1622, 1625, 1630, 1637, 1646, 1657, 1663,
1729, 1730, 1733, 1738, 1745, 1754, 1765, 1771,
1837, 1838, 1841, 1846, 1853, 1862, 1873, 1879
*ELSET, ELSET=S1
2559, 2560, 2561, 2562, 2563, 2569, 2575, 2581,
2587, 2593, 2599, 2605, 2611, 2617, 2623, 2629,
2635, 2641, 2647, 2653, 2679, 2680, 2681, 2682,
2683, 2689, 2695, 2701, 2707, 2713, 2719, 2725,
.
.
.
4363, 4369, 4375, 4381, 4387, 4461, 4462, 4463,
4464, 4465, 4471, 4477, 4483, 4489, 4495, 4501,
4507, 4513, 4519, 4525, 4599, 4600, 4601, 4602,
4603, 4609, 4615, 4621, 4627, 4633, 4639, 4645,
4651, 4725, 4726, 4727, 4728, 4729, 4735, 4741,
4747, 4753, 4759, 4765
**
**
** ForceX
**
*CLOAD, OP=NEW
FORCEX, 1, 1.
**
** F1
**
*DLOAD, OP=NEW
F1, P1, 1.
**
** S1
**
*DLOAD, OP=NEW
S1, P5, 1.
**
*TEMPERATURE, OP=NEW
**
*NODE PRINT, FREQ=1
U,
*NODE FILE, FREQ=1
U,
**
*EL PRINT, POS=INTEG, FREQ=1
S,
E,
*EL FILE, DIR=YES, POS=INTEG, FREQ=1
S,
E,
**
*EL PRINT, POS=NODES, FREQ=0
**
*EL FILE, DIR=YES, POS=NODES, FREQ=0
**
*EL PRINT, POS=CENTR, FREQ=0
**
*EL FILE, DIR=YES, POS=CENTR, FREQ=0
**

```

```

*EL PRINT, POS=AVERAGE, FREQ=0
**
*EL FILE, POS=AVERAGE, FREQ=0
**
*MODAL PRINT, FREQ=99999
**
*MODAL FILE, FREQ=99999
**
*ENERGY PRINT, FREQ=0
**
*ENERGY FILE, FREQ=0
**
*PRINT, FREQ=1
**
*END STEP
*STEP, NLGEOM, INC=50
**
**
This load case is the default load case that always appears
**
*STATIC
**
**
**
** ForceX
**
*CLOAD, OP=NEW
FORCEX, 1,      50.
**
** F1
**
*DLOAD, OP=NEW
F1, P1,      1.
**
** S1
**
*DLOAD, OP=NEW
S1, P5,      1.
**
*TEMPERATURE, OP=NEW
**
*NODE PRINT, FREQ=1
U,
*NODE FILE, FREQ=1
U,
**
*EL PRINT, POS=INTEG, FREQ=1
S,
E,
*EL FILE, DIR=YES, POS=INTEG, FREQ=1
S,
E,
**
*EL PRINT, POS=NODES, FREQ=0
**
*EL FILE, DIR=YES, POS=NODES, FREQ=0
**

```

```
*EL PRINT, POS=CENTR, FREQ=0
**
*EL FILE, DIR=YES, POS=CENTR, FREQ=0
**
*EL PRINT, POS=AVERAGE, FREQ=0
**
*EL FILE, POS=AVERAGE, FREQ=0
**
*MODAL PRINT, FREQ=99999
**
*MODAL FILE, FREQ=99999
**
*ENERGY PRINT, FREQ=0
**
*ENERGY FILE, FREQ=0
**
*PRINT, FREQ=1
**
*END STEP
```

REFERENCES

1. Litvin, F.L., Egelja, A., Tan, J., Chen, D.Y.-D., and Heath, G.F.: "Handbook on Face Gear Drives With a Spur Involute Pinion," NASA/CR-2000-209909, ARL-CR-447, March 2000.
2. Litvin, F.L., Wang, J.C., Bossler, R.B., Chen, D.Y.-D., Heath, G.F., and Lewicki, D.G.: "Face Gear Drives: Design, Analysis and Testing for Helicopter Transmission Applications," NASA TM-106101, AVSCOM Technical Report TR 92-C-009, 1992.
3. Hefeng, B., Savage, M., and Knorr, R.J.: "Computer Modeling of Rack-Generated Spur Gears," Mechanism and Machine Theory, Vol. 20, No. 4, pp. 351-360, 1985.
4. Handschuh, R.F. and Bibel, G.D.: "Comparison of Experimental and Analytical Tooth Bending Stress of Aerospace Spiral Bevel Gears," Journal of Mechanical Design, Vol. 121, No. 4, pp. 565-578, 1999.



Figure 1.—Plot of all points from Fcgr.exe. The fillet on both ends is seen to wrap around onto the fillet of adjacent tooth.

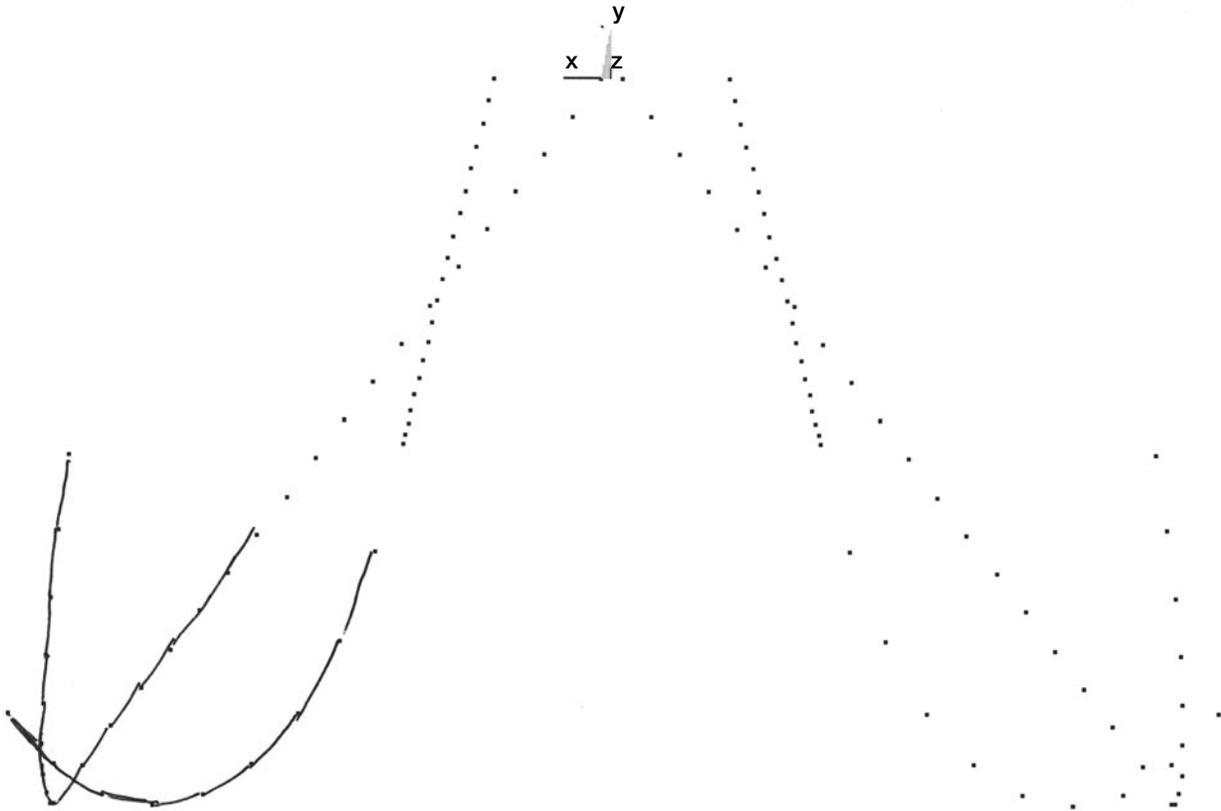


Figure 2.—Plot of points from Fcgr.exe on the inner most and outer most sections. Points above the bottom of the fillet are wrapped onto the next tooth.

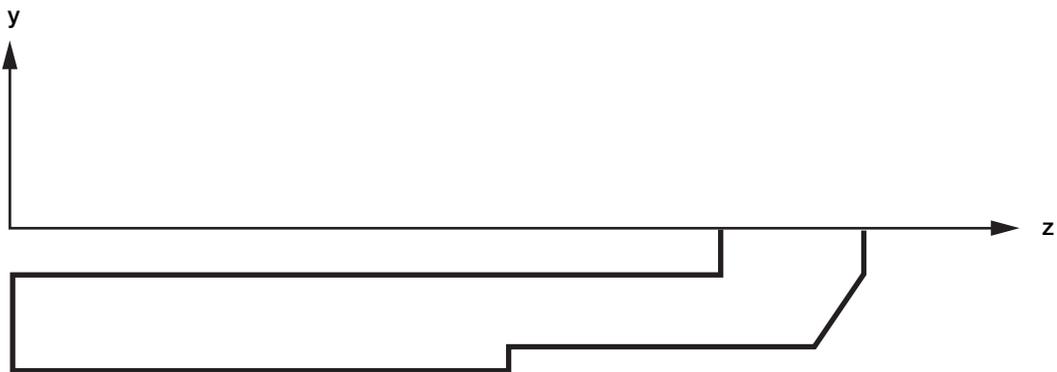


Figure 3.—Cross section of face gear. The y axis is the axis of rotation. All points on the face gear from the Fortran program Fcgr.exe start at $y = 0$.

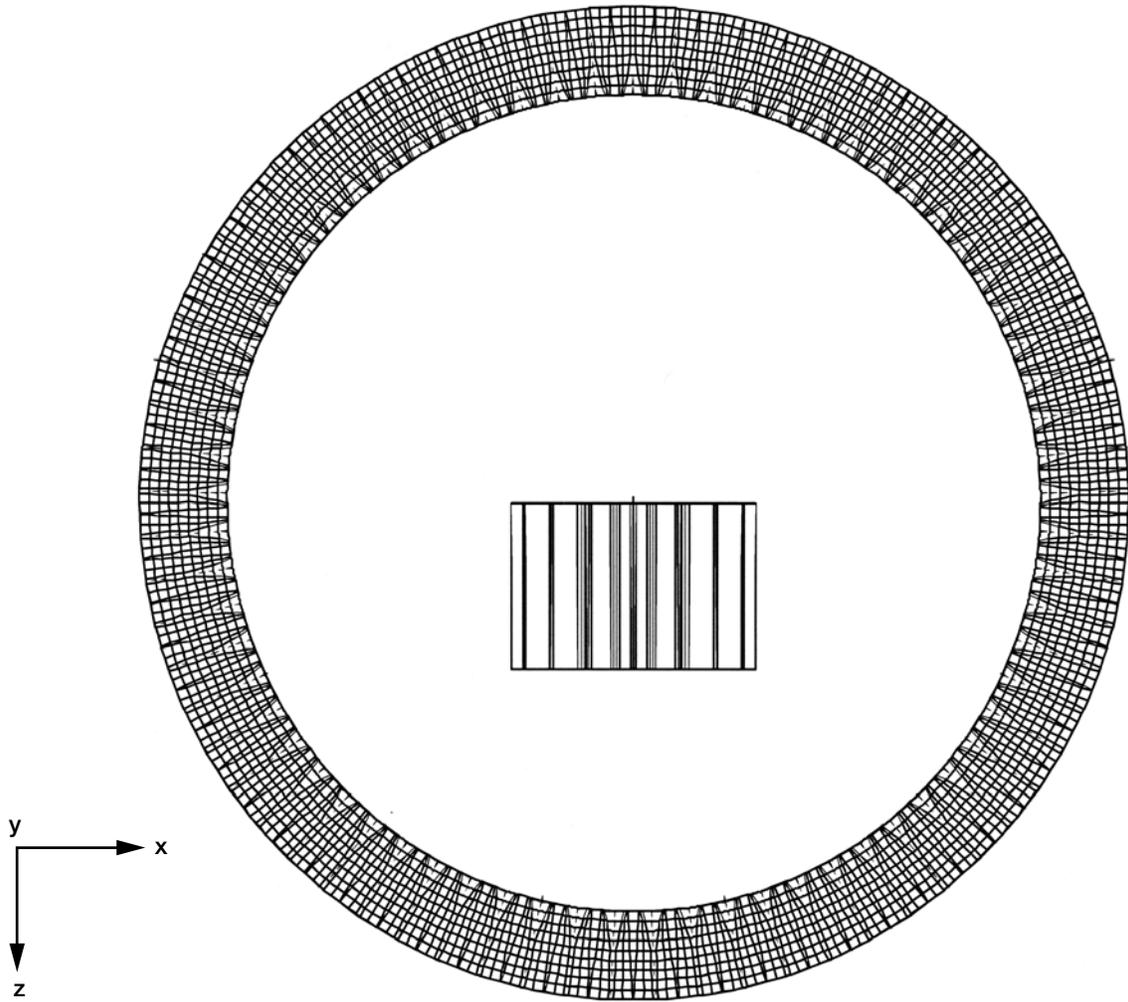


Figure 4.—Face gear is in xz plane. Spur gear is in xy plane. The 2D spurgear profile was extruded 1.285 in. in +z direction to establish depth of spur gear.

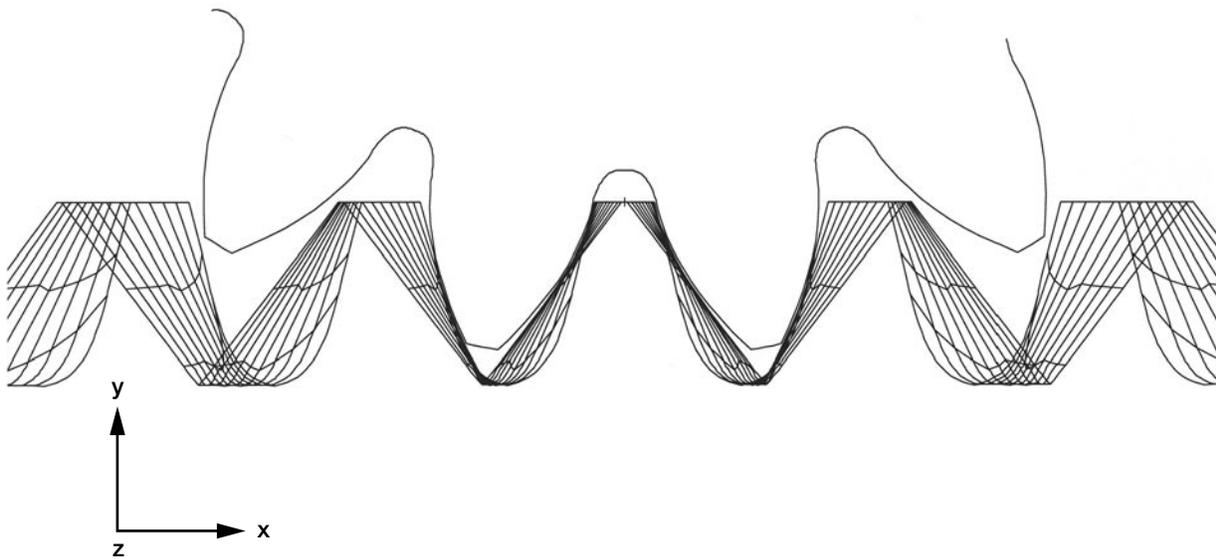


Figure 5.—Patran zoom on this figure verifies meshing without interference.

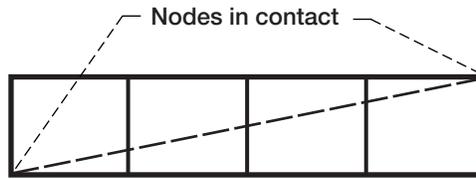


Figure 6.—Contact occurs diagonally across a row of elements indicating mesh density needs to be increased.

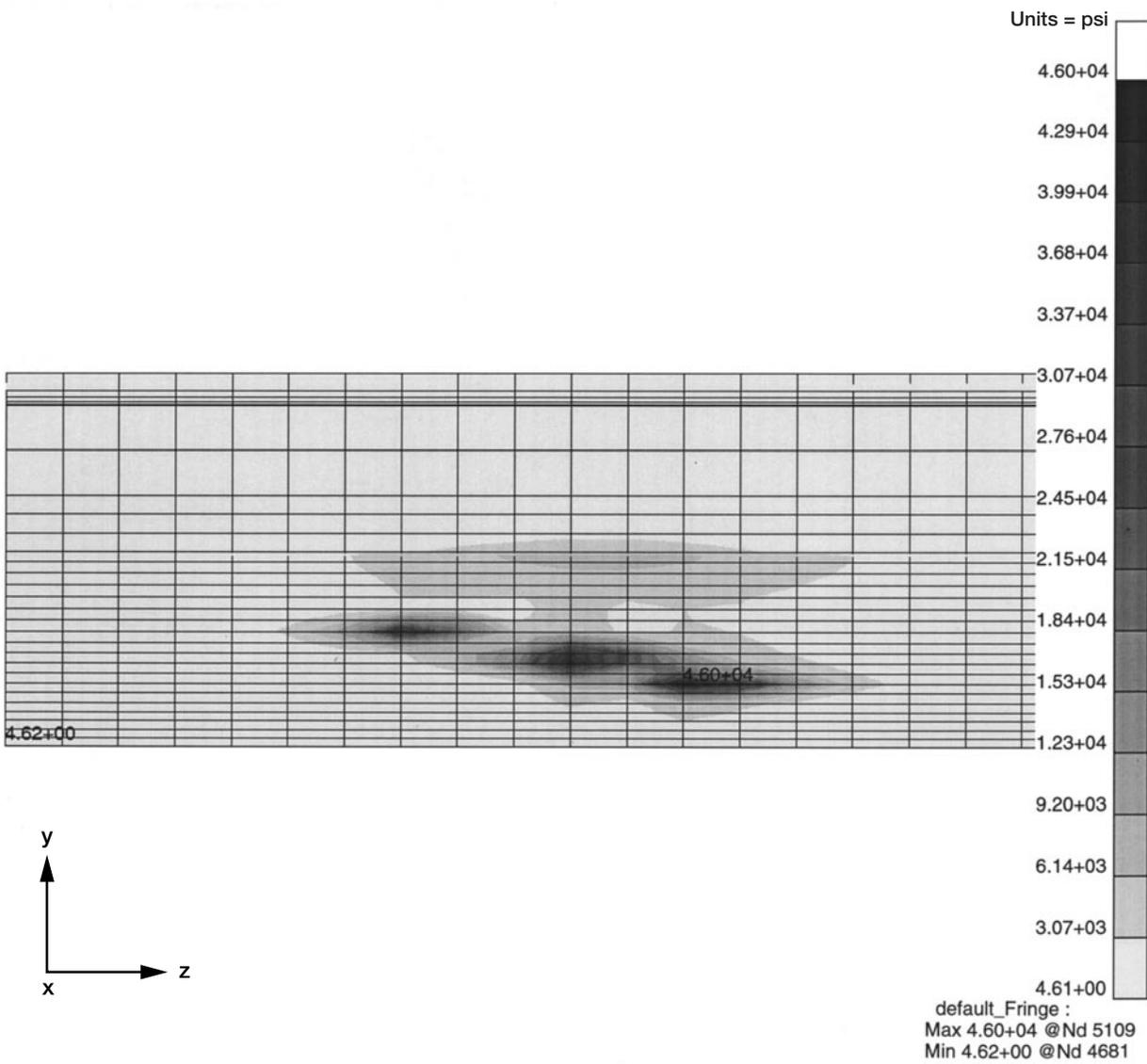


Figure 7.—Contact stress contours on the spur gear.

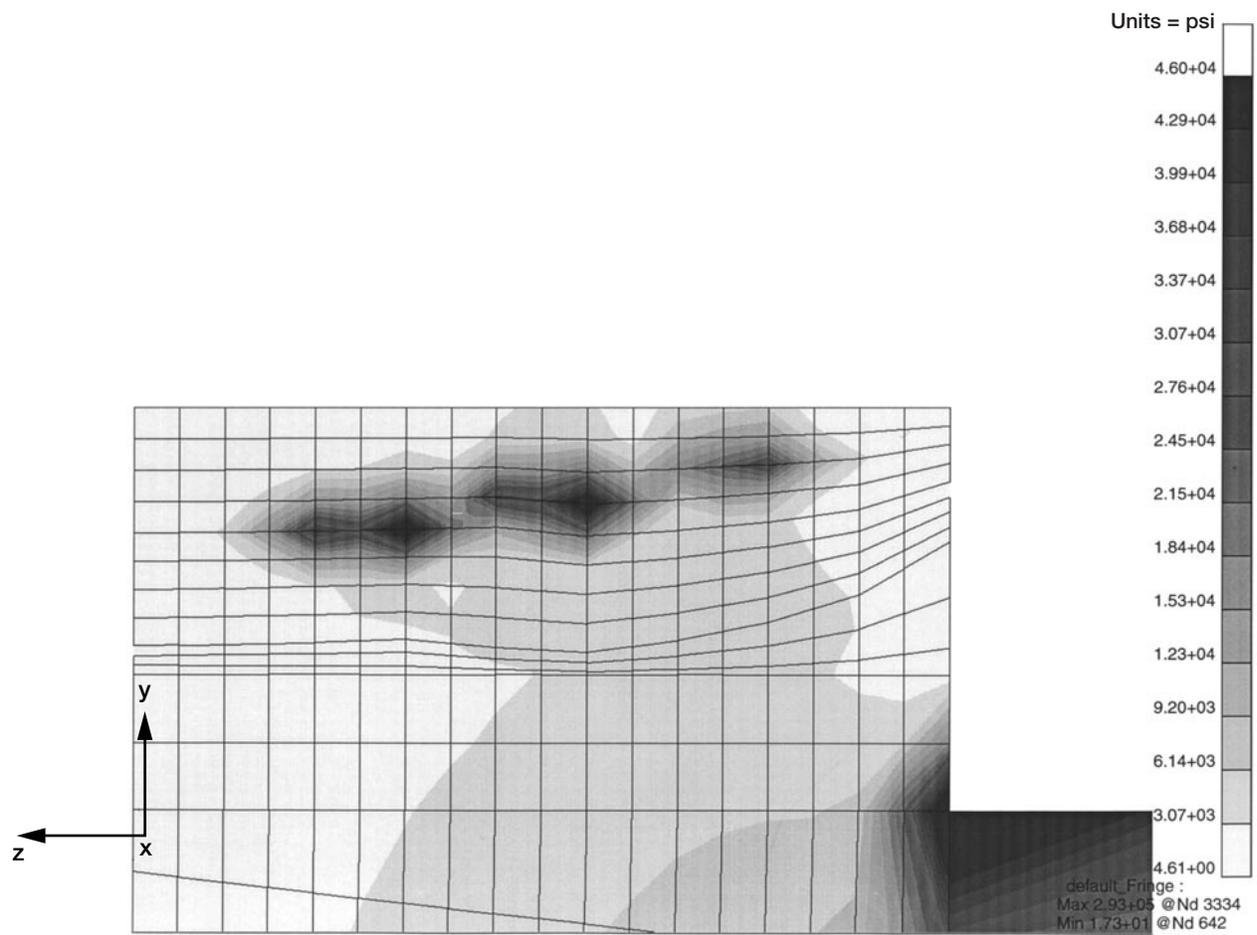


Figure 8.—Contact stress contours on the face gear.

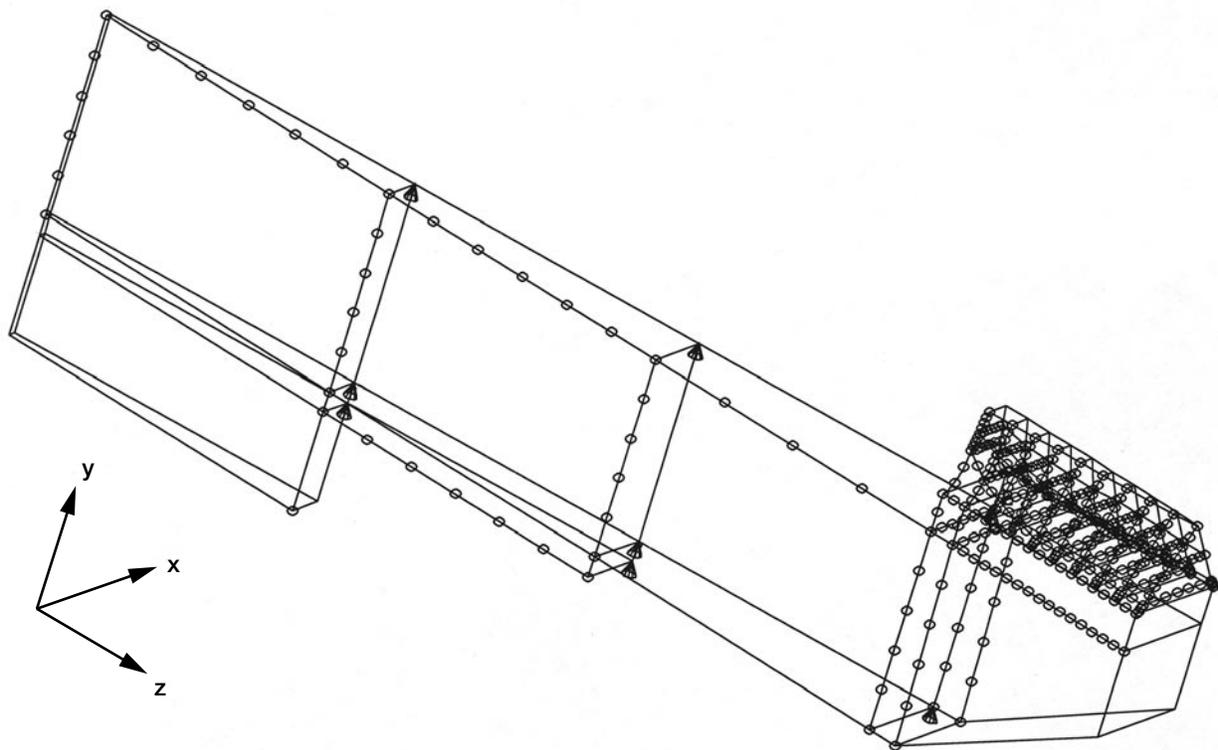


Figure 9.—Mesh seeds used on face gear rim.

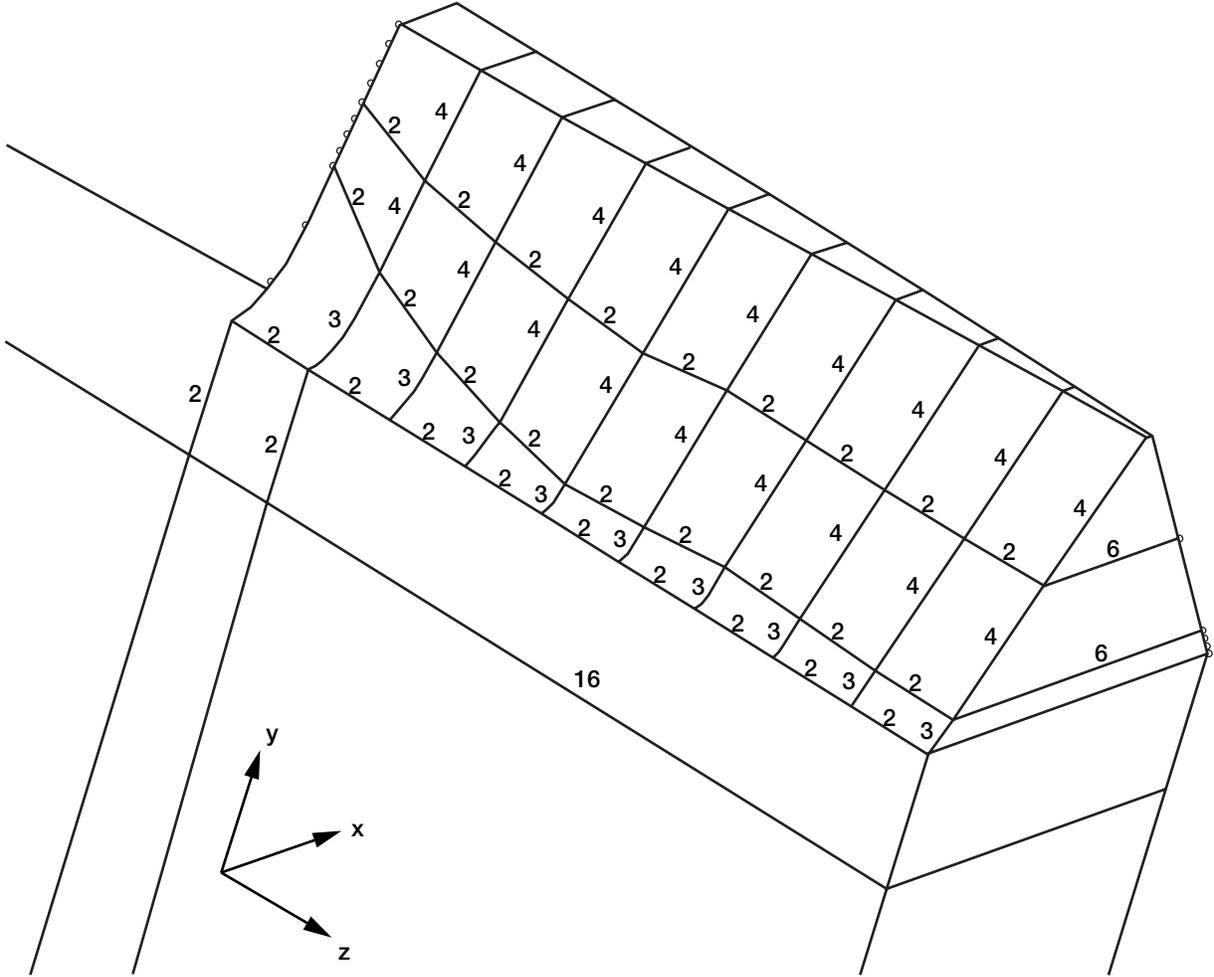


Figure 10.—Mesh seeds used on face gear tooth.

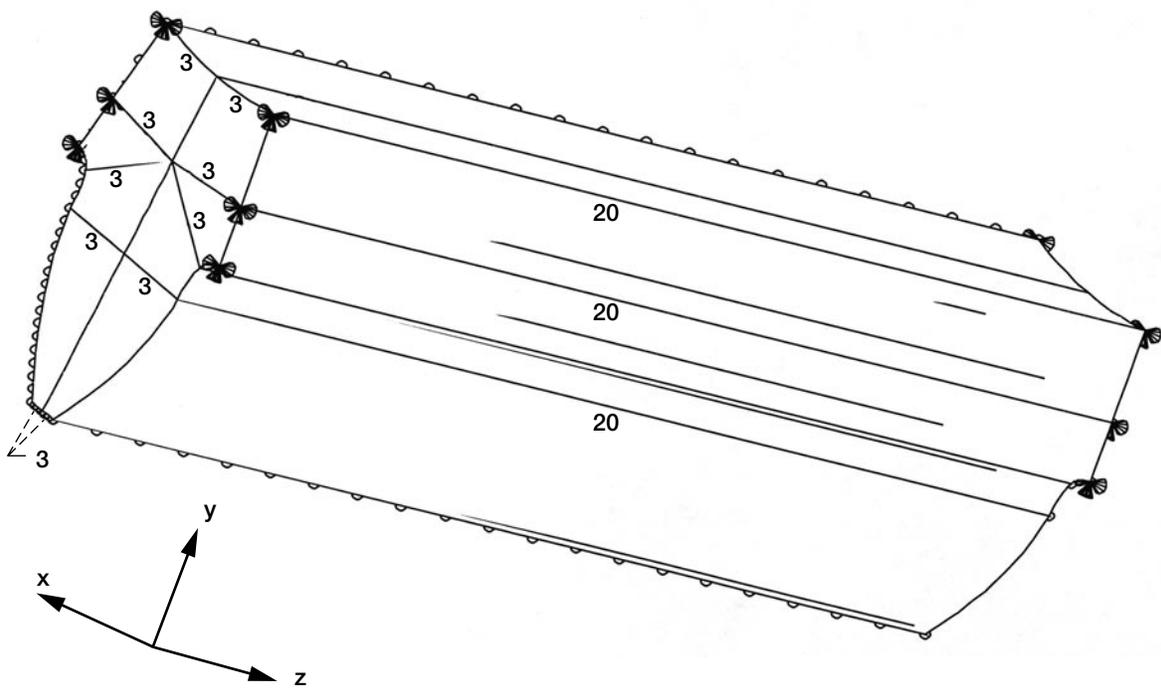


Figure 11.—Mesh seeds used on spur gear.

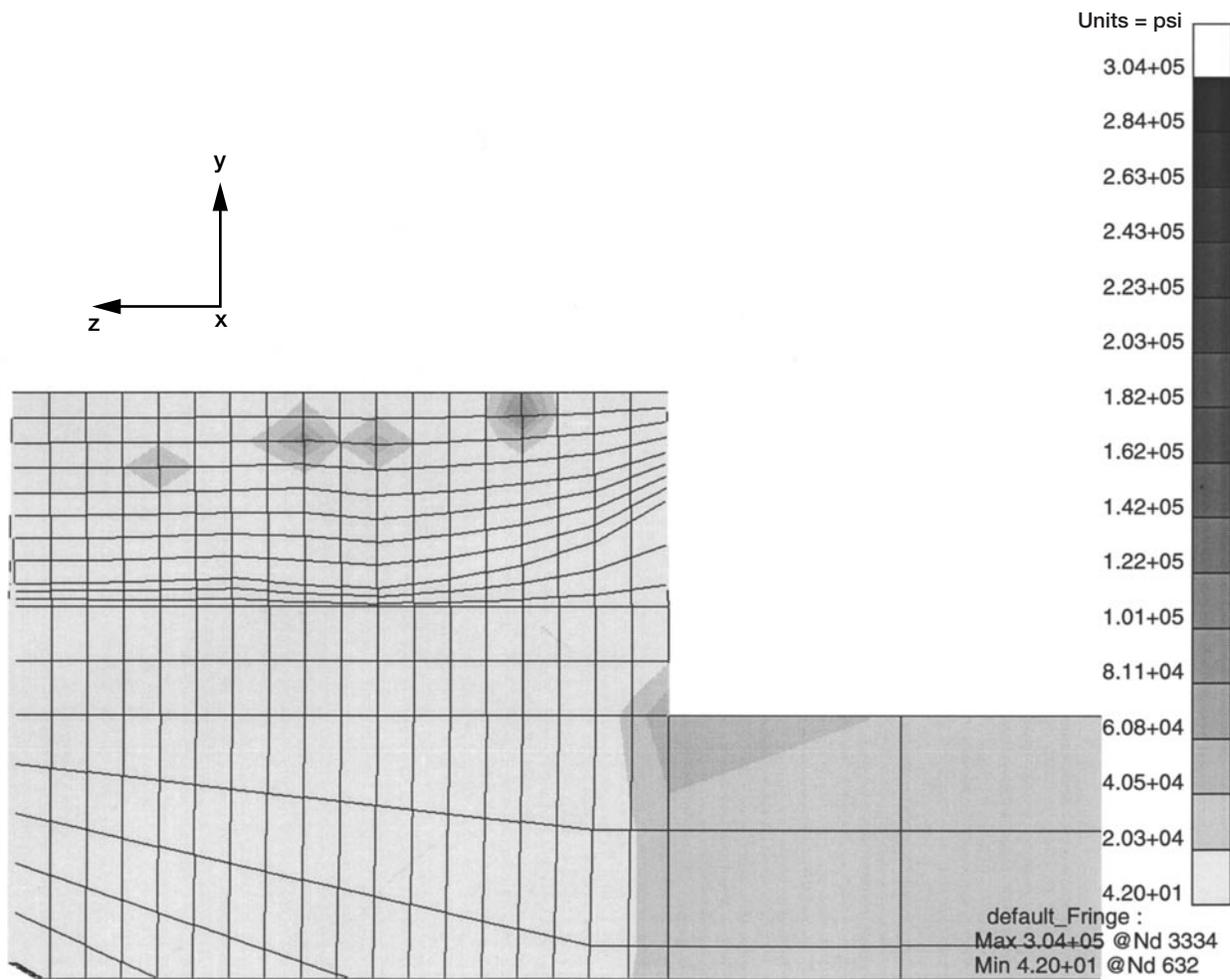


Figure 12.—Show contact pattern shift after rotation of gears accomplished by rotating local coordinate system. Compare to contact pattern in Figure 8 to verify rotation of contact pattern.

REPORT DOCUMENTATION PAGE			<i>Form Approved</i> <i>OMB No. 0704-0188</i>	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE January 2002	3. REPORT TYPE AND DATES COVERED Final Contractor Report	
4. TITLE AND SUBTITLE Procedure for Tooth Contact Analysis of a Face Gear Meshing With a Spur Gear Using Finite Element Analysis			5. FUNDING NUMBERS WU-712-30-13-00 NCC3-904	
6. AUTHOR(S) George Bibel				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) University of North Dakota Department of Mechanical Engineering Grand Forks, North Dakota 58202			8. PERFORMING ORGANIZATION REPORT NUMBER E-13076	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA CR-2002-211277	
11. SUPPLEMENTARY NOTES Project Manager, David G. Lewicki, Structures and Acoustics Division, organization code 5950, 216-433-3970.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category: 37 Available electronically at http://gltrs.grc.nasa.gov/GLTRS This publication is available from the NASA Center for AeroSpace Information, 301-621-0390.			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) A procedure was developed to perform tooth contact analysis between a face gear meshing with a spur pinion using finite element analysis. The face gear surface points from a previous analysis were used to create a connected tooth solid model without gaps or overlaps. The face gear surface points were used to create a five tooth face gear Patran model (with rim) using Patran PCL commands. These commands were saved in a series of session files suitable for Patran input. A four tooth spur gear that meshes with the face gear was designed and constructed with Patran PCL commands. These commands were also saved in a session files suitable for Patran input. The orientation of the spur gear required for meshing with the face gear was determined. The required rotations and translations are described and built into the session file for the spur gear. The Abaqus commands for three-dimensional meshing were determined and verified for a simplified model containing one spur tooth and one face gear tooth. The boundary conditions, loads, and weak spring constraints were determined to make the simplified model work. The load steps and load increments to establish contact and obtain a realistic load was determined for the simplified two tooth model. Contact patterns give some insight into required mesh density. Building the two gears in two different local coordinate systems and rotating the local coordinate systems was verified as an easy way to roll the gearset through mesh. Due to limitation of swap space, disk space and time constraints of the summer period, the larger model was not completed.				
14. SUBJECT TERMS Gears; Transmission; Machine elements; Finite element analysis			15. NUMBER OF PAGES 30	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	