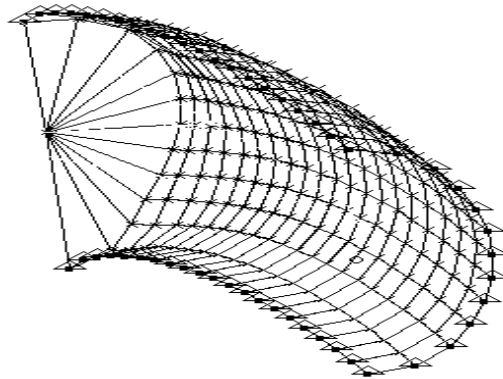# Linear Correction of Buckled Panels using SOL 200

Abstract

A method to account for the buckling of skin panels on stiffened shell structures using MSC/NASTRAN SOL 200 linear static optimization is shown. This builds on previous work, shows conservatism of the classical manual methods, and introduces several ways to tackle the problem. The Finite Element Analysis (FEA) involves the comparison of the use of Design Optimization (SOL 200) for effective width and thickness changes, to example A20.4 of Bruhn's text[1]. The Finite Element model uses a small fuselage section with a height of 50 inches, width of 30 inches. The foundation work for crippling strength calculation software is provided in an Appendix. This work is applicable to curved shell buckling issues in its use of empirical data available in the literature. This method's best implementation will be with p-elements due to their geometry information. The method has application to general stiffened curved panel structural analysis.

Michael C Kobold, PE

kobolmi@mail.northgrum.com, (407) 951-6829,
Northrop Grumman Corporation, Electronic Systems Integration Division, 2000 NASA Boulevard, P O Box 9650, Melbourne, Florida 32950-9650
This work is a donation to Actual Analysis, a non-profit corporation, www.absolute.org,
mkobold@absolute.org, (407) 720-6530, 768-0923
P O Box 372176, Satellite Beach, FL 32937-0176

---

[1]   Bruhn E F, MS CE Dr Eng, Professor Emeritus of Aeronautics and Astronautics, Purdue U., *Analysis and Design of Flight Vehicle Structures*, Jacobs Publishing, 1973, (317) 844-9400.

Introduction

The purpose of this procedure (herein called "opti-buckling") is to develop an easy method to correct linear static fuselage Finite Element models for buckled sheet developed during limit loads[2]. This paper uses a more neutral description, maximum service loads. Studies in the 1940's facilitated reduction of mass in aircraft such that they operated with the skin no longer shear resistant, but at reduced compressive effectiveness (thinner) with stiffening members taking much of the bending load[3]. At maximum service bending the compressive side of a fuselage will load its panels past their buckling strength, $_{cr}$. Here we use the terminology of Bruhn[4], $_{cr}$, since this paper also uses his example A20.4 to compare his hand calculation results to the results of a linear MSC/NASTRAN Sol 200 iteration solution, manual optimization study, and a separate nonlinear Sol 106 analysis.

A previous paper[5] Steven Basic presented at the 1995 MSC World Users' Conference discussed the general capabilities of the use of MSC/NASTRAN for stiffened shell structures and fundamental theory of compressed shells.

The objective of this system is to merge various historic hand calculation methods[6] with MSC/NASTRAN's optimization SOLution Sequence 200 (SOL 200) for answers in about three iterations. The manual calculations shown in texts by Bruhn[2], Peery[7], Niu[8], and others[9] were in use since long before[10] the Turner constant strain finite element triangle[11] came into service in 1957. These manual calculations use empirical buckled panel data and assume elasto-plastic behavior for a portion of the section (that we will call $A_{ineffective}$) – all this work going on long before computer assisted FEA arrived[11] in 1957. Another objective of this paper is to provide access to templates allowing users to ramp up the learning curve and use opti-buckling on site within a day or two. The use of optimization in FEA is usually not trivial. The templates such as http://www.absolute.org/optibuckl01.txt may also be available from the MSC site[12].

The majority of this paper deals with a manual optimization using a succession of MSC/NASTRAN SOL 101 linear static runs to validate the idea. The scripts for SOL200 will be available at www.absolute.org/amuc98/optibuckle.html.

This method is not only expandable to p-elements, it may best be used with p-elements for more accurate curvature and arc length measurement of panels. One of the advantages of the historical (herein called "Bruhn"[13]) method iteration method is that for a given material the critical buckling strength, $_{critical}$, is a function of geometry *only*. Curvature, r, and arc length, b, are input for $_{cr}$. In the 2nd example we manually input r, b, and $_{cr}$ to test the optimization method. A program or pcl code can query h-element

---

[2]   There is some strong willed controversy over the description of aircraft loading leading to a difference in jargon used in different organizations. E F Bruhn uses the terms "limit" and "applied" interchangeably as shown on page A4.1 of *Analysis and Design of Flight Vehicle Structures*, 1973. Bruhn also uses the terms "ultimate" and "design" to refer to the same loads, these are both equal to the the applied load * Factor of Safety.Fracture Mechanics folks such as David Broek use the term "ultimate" to mean "design", the stress at which the structure fails catastrophically, whereas the "service" load is the "applied" load, on page 4 of *The Practical Use of Fracture Mechanics*, Kluwer Academic Publishers, 1989. In order to calculate the residual strength we need the maximum service load the structure will usually experience.
[3]   MELCON, M.A., ENSRUD, A.F., Lockheed Aircraft Corporation, "*Analysis of Stiffened Curved Panels Under Shear and Compression*," Journal of the Aeronautical Sciences, February 1953, pp. 111-126. See the "Post Buckling Behavior of a Shear Panel" section.
[4]   Bruhn E F, MS CE Dr Eng, Professor Emeritus of Aeronautics and Astronautics, Purdue U., *Analysis and Design of Flight Vehicle Structures*, Jacobs Publishing, 1973, (317) 844-9400.
[5]   Basic S., *A Design Stage Non-Linear Idealization of Stiffened-Compression Panels for Linear MSC/NASTRAN/ARIES Modelling of Diagonal Tension Field Shells*, MSC 1995 World Users' Conference Proceedings, Number 30.
[6]   MIL-HDBK-5G reference 1.7.1(b), Gerard and H Becker, *Handbook of Structural Stability*, US National Advisory Committee for Aeronautics, Technical Note, Numbers 3781, 102 pp (July 1957), 3782, 72 pp (July 1957); 3783, 154 pp (August 1957); 3784, 93 pp (August 1957); 3785, 89 pp (August 1957)
[7]   Peery, David J, PhD, and Azar, J J, PhD, *Aircraft Structures*, McGraw-Hill Book Company, 1982
[8]   Niu, Michael C Y, *Airframe Structural Design*, 1988, *Airframe Stress and Sizing*, 1997, Conmilit Press, Ltd., Hong Kong, (310) 475-5711.
[9]   Flabel, Jean-Claude, *Practical Stress Analysis for Design Engineers*, Lake City Publishing Company, 1997, page 540 ff.
[10]   MIL-HDBK-5G reference 1.7.1(a), *Index of Aircraft Structures Research Reports*, US National Advisory Committee for Aeronautics, Index Number 7E29, 40 pp (June 1947)
[11]   TURNER, CLOUGH, MARTIN, and TOPP, "*Stiffness and Deflection Analysis of Complex Structures,*" Journal of the Aeronautical Sciences, Vol. 23, September 1956, pp.805-823.
[12]   http://www.macsch.com/aerospace.html
[13]   Just to reference his book's description of it.

mesh connectivities to get an average r and use this curvature to calculate b based on the connectivity. Structure charts are available to speed writing of this program, as described later.

This paper focuses on cylindrical airframes with several different curvatures, but the method is far more general, especially if used with p-elements. The proposed curvcritstress program determines curvature in 2 dimensions. The curvcritstress program can give sufficient accuracy for h-mesh models. Due to the accuracy and availability of geometry in p-elements, a p-element model could create a more automated and accurate opti-buckling system.

Depending on the overall scale, airframe FE models have evolved towards the use of CQUAD4 elements rather than CSHEAR elements to model the aircraft skin. CSHEAR elements *require* surrounding stiffeners. But the CQUAD4 more accurately accounts for membrane, bending, and shear stiffness[14], and in combination with many different stiffener models is superior to the old CSHEAR methods[15]. However, old hands like the CSHEAR FE models because they lump panel section areas into stiffeners just like the manual calculations, and so are easier to compare to manual calculations of details.

For most aircraft, shear in the skin is not in the fully diagonal tension regime. There is some compressive strength up to a point, so the skin shear is in the semi-diagonal tension region[5]. Unfortunately, the CQUAD4 puts us in the shear resistant regime since its shear stiffness is not a function of load. Use of doubled coincident elements with orthogonally oriented orthotropic material properties is a possible automated solution to modeling semi-diagonal tension. Although related to this paper's buckling issues, diagonal tension is another field in and of itself.

Methods developed and taught by Bruhn, Peery, Niu and other aerospace leaders allow aerospace "stress" engineers[16] to calculate airframe stresses quickly, often faster than with FEA. These manual methods usually involve assuming lumped area models. So shear calculations use only shear flows and areas rather than a calculation or use of a tabulated value of Q. However, looking at just the P's and A's (lumped loads and lumped areas) does not account for many things including variation of in-plane shear stress over the section. When we use Finite Elements in an FE model, we get the increase in shear distribution accuracy and precision for "free" by use of the opti-buckling procedure shown in this paper, as opposed to purely manual calculations. Finer meshes can account for local rivet pitch – discreet load transfer from panels to stiffeners[17] and vice versa.

The manual methods (Bruhn, et al) use a combination of structures theory and empirical evidence to calculate various strengths and parameters. This paper uses the strength $_{cr}$ for bending related to membrane compression only. Full manual calculation of $_{cr}$ involves combinations of pressurization, shear, torsion, bending, and axial loading[18]. MSC/NASTRAN's SOL 200, the design optimization feature, can provide the same calculation for use with linear static FE models so prevalent in preliminary and refurbishment analysis today.

The Manual Method: In what we will call "Bruhn's iteration method" the engineer calculates the $_{cr}$ based on panel curvature. Using this value $_{cr}$ 0.3 Et/r in psi, the engineer corrects the loading in the panel undergoing compression. We will add some of the other modifications to $_{cr}$ in the later portion of this work for Pressure and Shear Effects[18]. The load application part of this analysis is "free" since this is an FE model. The compression correction (buckled sheet "effective area" or "effective width" discussed

---

[14]   "One reason for the switch to quads is that by using quad elements to idealize web structure, you get an accurate representation of web in-plane shear, axial and bending stiffness without having to lump web areas into adjacent chord areas. Using shear elements to idealize the web of a frame for instance, you cannot obtain accurate frame bending and axial stiffnesses at the same time. Using QUAD elements for this type of idealization better represents the actual structure." posits John Kummer.

[15]   This discussion is beyond this paper and methods vary across organizations. See reference 5.

[16]   In some other industries these folks would be "analysts" of an Engineering Analysis group. In airframe engineering the non-FEA folks in the stress group often outnumber the FE analysts (often called FE modelers though they do FEA). This opti-buckling method should remove one of the barriers to use of FE models by many stress engineers.

[17]   Including for most of our examples, stringers, but also longerons, and box beam stiffeners and beam caps.

[18]   In the A20.4 example this paper follows, Bruhn uses $_{cr}$ = 0.3 Et/r as a conservative value for pure bending generated membrane stress strength. Later in chapter C9 Bruhn introduces most of the other influences one should consider including pressure, torsion, and shears. Then section C11.32 completes these calculations.

later) is the major difference between the use of most linear static FE models and the often more accurate manual calculations using lumped structure. The manual methods include differential stiffness!

Caution: Bruhn's text uses the term effective *area*[19] for the section area of the panel that can only support $_{cr}$ in order to calculate its physical contribution to I_{yy}, the 2nd moment of area. But the effective *width*[20] term is that portion of the total panel cross-sectional area that is fully effective since it carries the full membrane stress, or $_{bending}$. This is the membrane stress at arm z, the location of the longitudinal stiffener (stringer) from the neutral axis of the vehicle as it acts like a beam in bending. In this paper we will use the term "ineffective area", A_{ineff} or A_i. This cross-sectional area will only carry the load $P_{ineff} = A_i {}_{cr}$.

The first iteration in the manual method assumes a uniform application of $_b$ (opti-buckling does the same). The buckled sheet carries only the amount of load determined by $_{cr}$ multiplied by the "ineffective area", A_i, plus the membrane stress, $_b$, times the "effective area", A_e. However, in the manual methods airframe stress engineers "lump" this effective area into the stringers as an "effective width" of panel[21]. This method allows the stress engineer to calculate a new neutral axis.

Opti-buckling also uses A_i and A_e. However, in the linear method we may want to reduce the section further in order to lower the stiffness of that panel so the new loads (element forces), $P_{ineff} + P_{eff}$, match the manual method's elasto-plastic nonlinear material calculation, where $P_{ineff} = A_i {}_{cr}$. This correction will quicken convergence. This is just bookkeeping. The manual method itself uses effective widths that are bookkeeping for load distribution. Opti-buckling uses SOL 200 or user input to do the same bookkeeping. The bookkeeping can take many forms. Overcompensate by reducing the panel thickness past the area calculations and let the optimization iterations correct for this (the method this paper covers), or use a parameter to increase the total load on the structure.

For the manual method, usually only a second iteration is necessary[22]. This is a motivation for opti-buckling development. Application of the new $_{cr}$, after accounting for the new neutral axis, gives a fairly accurate picture of the loads in the stiffening stringers or longerons and all the skin panels with just two iterations.

The Use of Optimization: Some manual calculations easily transform into a design variable. The effective area calculation becomes an optimization using element membrane stress as the design response for shell element thickness, the design variable. We shall use normal stress. We omit the transfer of cross-sectional area from the panels into the stringers (stiffeners), keeping w   1.7 t sqrt(E / $_b$) in the bookkeeping.

Since the cross-sectional area is a linear function of the shell element thickness, optimization of element thickness can reduce the effective area for those elements with compressive membrane stress. Since this FEA method (opti-buckling) uses a general multi-application software (MSC/NASTRAN), this optimization can occur in two different curvature dimensions at the same time, allowing analysis of pressure domes as well. The first automated version of this method calculates curvature of the structure even with flat CQUAD4 elements by query of connectivity to nearest neighbor elements (ccs_geom). Here p-elements will give us a bit more accuracy.

Those of us who expect that the "h-method" will someday catch up with and surpass the "p-method" by use of user-interactive software for FE model meshing decisions should look at this use for p-elements with opti-buckling. The p-elements are ready-made for getting the necessary input for these critical buckling

---

[19]   Bruhn's method uses a partition of the cross-sectional area into two cross -- sectional "areas": an elasto-plastic area, and an area that remains forever linear elastic. A wealth of test data in Aerospace comprises tabulations of areas, or their ratios, for various sheet metal curvatures and loading.

[20]   Also described in *Airframe Structural Design*, 1988, pages 142 ff, Michael C Y Niu, Conmilit Press, Ltd., Hong Kong, (310) 475-5711.

[21]   As mentioned in the "Caution:" paragraph above.

[22]   Bruhn, Reference 1 or 4, Page A20.6, " . . . If a third trial were used, . . . differences would be quite small. . ."

strengths. Additionally we get more accurate curved panel stresses[23]. At some point these advantages outweigh the libraries of flat plate comparisons to test. In the near future, the user of p-code need not use an external program to calculate the curved sheet critical buckling strength because all the necessary data is already contained in the geometry (connectivity), property, and material entries for a p-element, and standard buckling data[24]. The main improvement is reduction in the number of iterations to a known small number, and the increase in accuracy due to the use of existing empirical data.

Even for h-meshes there is little or no extra input. Compare input requirements for opti-buckling to those for nonlinear buckling. For brand new structures, analysis does not always assume the nonlinearity is just geometric, it can include material nonlinearity unless test indicates otherwise (CPU time is cheap). So, rather than input a stress – strain curve we ordinarily use with "severe" buckling analysis, the opti-buckling user inputs geometry parameters[25] and critical strengths, or uses a program such as ccs to do this as described later.

For automated h-mesh curvature averaging, how does the nearest neighbor curvature calculation compare to the actual curvature of the mathematically ideal model that the FE model seeks to represent? There is an error in reported curvature due to averaging which is essentially an error in calculation of the moment arm to the center of a flat element representing the curved sheet. These details are discussed in the Accuracy in Modeling section.

The first opti-buckling iteration assumes, as does Bruhn's A20.4 manual iteration, that the panels are all fully effective. The design response, panel membrane stress, will be the input for a first iteration of panel thickness "optimization". MSC/NASTRAN gives us extra iterations easily. The reduction of sensitivity to panel thickness stops the Sol 200 run in MSC/NASTRAN. This compares to the negligible changes in neutral axis location we see in the manual method (see A20.4 Trial 2 and 3 Results).

Results in this paper use manual entry of the ccs output to validate the method. Comparisons between results from the manual method, linear opti-buckling manual iteration, Sol 200 opti-buckling, and Sol 106 non-linear analysis validate this method.

Problem Definition

<u>Critical Buckling Strength:</u>   There are many sources for calculation of critical buckling strengths for combinations of loads including longitudinal compression (and vehicle bending), shear, and internal / external pressure[26]. The value $_{cr}$ used in example A20.4 of Bruhn's book is conservative, used only for bending stress alone, and found in other references[27,28]. The Appendices of this paper document the algorithm for a program for critical buckling strength generation called Curved Critical Strength, ccs. But hand calculation of $_{cr}$ is sufficient. The first part of the program, ccs_geom, calculates the radii and panel widths. The second part of the curvcritstress program, ccs_loadsum, uses this geometry from the first part to calculate the critical buckling strengths.

The curvcritstress program first determines curvature in   –   space (xi - zeta like the old Patran surface patch orientation). Then the program decides whether curvatures are reasonable. It considers large curvature as a property of an essentially flat plate. If both dimensions have curvature within $100\,t$ to $10^4\,t$ then the

[23]   Patel S, Hoff C, and Gwillim M, *Local Stress Analysis of Stiffened Shells Using MSC/NASTRAN's Shell and Beam p-Elements*, 1997 MSC Aerospace Users' Conference Proceedings

[24]   Bruhn, reference 1 or 4, Chapter C11.32.

[25]   We manually enter radius and width b here, but the automated version, ccs_geom, automatically generates this data.

[26]   In addition to Bruhn, Niu, Peery, and others, some of this data is in the NASA Astronautics Structures Manual N76-76166 Part 1 (Volume 1) NASA Technical Memo, NASA *TM X-73305*, George Marshall Space Flight Center, Alabama, Section A3, February 15, 1976, Page 27

[27]   Peery, David J, PhD, and Azar, J J, PhD, *Aircraft Structures*, McGraw-Hill Book Company, 1982, pages 364ff.

[28]   Niu, Michael C Y, *Airframe Stress and Sizing*, 1997, Conmilit Press, Ltd., Hong Kong, (310) 475-5711, pages 586 ff,.

system saves the data for use with doubly curved or spherical (r_1 within 5% of r_2) values of $_{cr}$. If both dimensions are large, the element is "flat" and removed from the optimized element set. Otherwise, the program reports the curvature and orientation of the curved edge to the second part of curvcritstress, ccs_loadsum. If any curvature is less than 300 thicknesses curvcritstress produces a warning (any curvature less than 100 thicknesses produces a fatal error). The curvcritstress program can give sufficient accuracy for h-mesh models. Due to the accuracy and availability of geometry in p-elements, a p-element opti-buckling system can be more automated and accurate.

Before entering the second (ccs_loadsum) part of curvcritstress, the program then uses the curvature to calculate the arc length, b, used in determination of effective area. Structure charts for these algorithms are in Addendum A.

In section C9.13 *Calculation of General Instability* of Bruhn's book[29], he determines the effect of pressure, transverse shear, torsion, and bending. Bruhn's section C11.32 completes this subject. The second part of the curvcritstress program, ccs_loadsum, will use these relations.

Effective Area, Bruhn's Method: Bruhn uses effective stringer width, w, and panel width, b, to calculate section areas. We use a variant of Bruhn's definition of effective area, A_e, to determine the design thicknesses. Bruhn uses the term "effective width" to determine that portion of the panel cross-sectional area that is fully effective, the area supporting the full $_{bending}$ of membrane stress. However, his A_e is an "effective area" that we will call ineffective area, A_{ineff}, or A_i, since it is that part of the cross-sectional area that only carries $_{cr}$ of stress (A_{ineff} is the elasto-plastic portion)[30]. The use of a thickness correction, rather than merely changing w per iteration, is the essential difference between the opti-buckling method and Bruhn's hand calculation method. In order to reduce the load carried by the buckled skin Bruhn lowers the load carried by the panel by lumping a part of section area representing an effective width of skin, w, into adjacent stringers. The opti-buckling method merely decreases the thickness.

In order to clarify notation, this paper will label iterations for both Bruhn's manual method and the opti-buckling method with a pre-superscript (e.g. $^j w$, the jth iteration's calculation of effective width). The element number or panel number will be a pre-sub-script (e.g. $_k b$, the width of Finite Element # k). This should avoid confusion with and other notation such as partial derivatives.

Bruhn uses b' as the width of buckled sheet. b' = b - $_+ w/2$ - $_- w/2$ where $_+ w$ and $_- w$ represent effective widths attributable to the upper and lower stringers (stiffeners) of the panel. In example A20.4 Bruhn starts with $^0 w = 30t$, an initial effective width estimate of thirty thicknesses of skin. This is a common practice in the industry. Subsequent effective widths in A20.4 use the common Effective Width formula. With j as the iteration number,

$$^j w \; = \; ^{j=!0} w \; = \; 1.7(t)\, \mathrm{sqrt}(E/\,_b) \qquad \text{where} \qquad ^0 w = 30t$$

Where the membrane stress from vehicle bending stress is $_b = Ez/I$ calculated at the arm, z. That moment arm z measures from the stringer the width of skin will be lumped into, down to the neutral axis. So $^0 b' = b - 30t$, and $^0 A_i = bt - 60t^2$. The first non-trivial iteration uses the effective widths calculated from the bending stress at the upper and lower stringers for a given panel. Use of pre-superscript j assumes j is not zero (j=!0).

$$^j_k b' \; = \; _k b - (^j_{k+} w/2) - (^j_{k-} w/2)$$

For this paper the effective area, A_e, is the cross-sectional area of the skin that carries the full compression stress. The ineffective area, A_i, only carries its critical (buckling) strength, $_{cr}$, and is therefore elasto-plastic. This point is essential for following the calculations. Bruhn's fully manual method is to segregate the cross-section into two zones, one fully effective and another that has buckled and can only carry $_{cr}$.

---

[29]   Bruhn, see reference 1 or 4.
[30]   See the "Caution" paragraph in the Introduction for clarification on our use of "effective".

The manual SOL 101 method used to validate opti-buckling uses a spreadsheet that tracks the effective and ineffective loads, and P_e and P_i. Tracking these loads per iteration we calculate the new skin thickness per iteration. The application of FEA requires correction for that portion of section that is elasto-plastic, shown in the spreadsheet[31] as the "design stress", a function of the "analysis stress" that MSC/NASTRAN outputs. However, the manual calculation method summarized here tracks this on paper.

The effective area is a function of the newly calculated stress through the effective width, w.

$$^j_k A_{eff} = {}_k bt - {}^j_k b't = {}_k bt - [{}_k b - ({}^j_{k+}w/2) - ({}^j_{k-}w/2)]t = [({}^j_{k+}w/2) + ({}^j_{k-}w/2)]\, t$$

$$^j_k A_{ineff} = {}^j_k b't = [{}_k b - ({}^j_{k+}w/2) - ({}^j_{k-}w/2)]t$$

The total load carried by the skin is important for the FEA iteration formulation.

$$^j_k P_e = {}^j_k P_i + {}^j_k P_e = ({}^j_k{}_{cr} * {}^j_k A_i) + ({}^j_k{}_b * {}^j_k A_e)$$

Here we have the first iteration using the first corrected effective widths. $^0 P_e = b\,{}^0 t\,{}^0{}_b$, $^0 P_i = 0$. We can turn these iteration formulae into thickness as a function of the effective *and* ineffective load calculations based on w. See the "Effective Area" section that follows. Notice that the new load for element k, iteration j, is a function of the stress from iteration (j-1) through the effective width calculation, $^j_k w \quad 1.7\ t\ \mathrm{sqrt}(E / {}^j_k{}_{cr})$.

CCS (curvecritstress program):

To calculate the correct critical buckling strength, $_{cr}$, the proposed program needs to check for curvature continuation so that a crease beam type curvature reversal is not averaged into a meaningless curvature. Structure charts[32] in Addendum A provide a look at the logic and features of the code. The availability of these structure charts should cut the cost of development in pcl, C, or Java significantly. Feel free to use them (Microsoft PowerPoint data available on request).
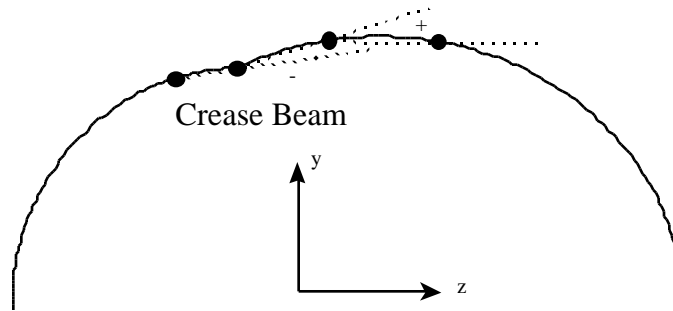

Figure 1: Curvature check for ccs_geom (Curve Critical Stress extraction program).

The second part of the ccs system, ccs_loadsum can generate the critical strength value per panel (per curvature value group) using information like that ccs_geom produces. ccs_loadsum uses input for pressurization, torsion, and shear as described in Chapter C11.32 of Bruhn. Then ccs_loadsum can generate the critical strength values, $_{cr}$, per curvature value group.

Effective Load Iteration for SOL 200: The first iteration of the MSC/NASTRAN based opti-buckling method starts, as does Bruhn's method, with $^0 w = 30t$. We saw how this gives us the total load

---

[31] See www.absolute.org/amuc98/effective4s.prn or effective4t.txt.
[32] To be posted to www.absolute.org/amuc98/ccs.html.

in our "design" model (not the "analysis" load FEA will produce). The next iterations use the w   1.7 t sqrt(E / ^j_k $_{cr}$) estimate.   We use the "analysis" stress from MSC/NASTRAN to calculate the "design" stress. From this "design" stress that includes the physical elasto-plastic nature of the buckled sheet, we can estimate the new thickness for the next iteration. The objective is assign only the critical strength to the ineffective section of the skin. After a few iterations this elasto-plastic effect is accounted for.

To help those unaccustomed with aerospace methods such as Bruhn's, a simple bar example, "Optimization Thickness, Four Bar Model Example," follows these derivations for thickness correction.

Usually optimization folks try to minimize a design variable, say $_{stress}$, to remain under some maximum design objective, say $S_{strength}$ (F for Aerospace folks). When   < S the world looks great, the design constraints are satisfied, no action is crucially necessary, but the optimizer shaves some weight (the design objective). When   > S the optimizer (SOL 200) can act to increase mass (increase t, I, et cetera) until < S to get back within the design constraints.

Opti-buckling acts similarly in a perverse way. This compressive strength, $_{cr}$, is a negative number. While the panel is fully effective,   > $_{cr}$, the world is fine. When the sheet is in tension (> 0) FEA is generally accurate, the linear model's neutral axis is not in error (due to buckling). When   0 >   > $_{cr}$ there is still no error due to buckling. However, when   < $_{cr}$ the panel in the mathematically ideal model is no longer fully effective, it is buckling. In reality buckling usually occurs before the linear SOL 105 results indicate because classical Euler buckling is under-conservative[33]. Aerospace engineering over the years developed the coefficients of these local panel instability relations empirically so this "theory" really uses experimental data. Other fields of engineering without these data can use extra factors of safety developed in testing. This would involve changing the coefficient 0.3 in $_{cr}$, 1.7 in w.

Ordinary optimization solutions counter the design objective when the FEA breaks out of or approaches the design constraints. When   < $_{cr}$ SOL 200 acts to reduce the panel thickness, ^j_kt, to reduce the A_{total} per the following opti-buckling relations derived from Bruhn's work. This accounts for the elasto-plastic effect of the b' width in the center of the skin panel.

$$^j_k A_{eff} =   b\, t - 1.7(t^2/2)\, \{\ sqrt(E/\ _{b+}) + sqrt(E/\ _{b-})\ \}   1.7\, t^2\, sqrt(E/\ _{ave}) = t\ w_{ave}$$

Where $_{ave}$ turns out to be NASTRAN's CQUAD4 centroid stress for a coarse loads model (the kind of model we want to use here) with one element per panel. The error induced by the use of $_{ave}$ is usually small as shown in the calculations in the Accuracy in Modeling section. Refinement to more than one shell (plate, etc.) element per skin bay[34] would be complicated and unnecessary for internal loads model. Calculation of the neutral axis location for opti-buckling using loads models (one shell/plate element per panel) is at least as accurate as historic manual fuselage stress analysis, except hand calculation does not account for warping of cross – sections. These FE models do account for section warp.

We need the value of panel width normal to the curvature axis, b, to calculate the new thickness. The automated system uses ccs_geom for this input. In the examples in this paper, we will enter the ccs_geom output data manually into NASTRAN.

The "**design**" load is a function of the "**analysis**" load, $^j_k P_{analysis} =   ^j_k\ _b * b * ^{j-1}_k t$.

---

[33]   Hence MSC often suggested engineers use nonlinear buckling via SOL 106 for even seemingly "simple" problems. Discussions by George Campbell from Ford about the WUC 1995 presentation, Schneider M, and Feldes of McDonnell Douglas Aerospace, and Dr. Claus Hoff, and Halcomb, J R, of MSC, *Stability Analysis of perfect and Imperfect Cylinders Using MSC/NASTRAN Linear and Nonlinear Buckling*, MSC 1995 World Users' Conference Proceedings, Number 29. This presentation itself is a great resource for these buckling issues.
[34]   For non-aerospace folks, a skin bay is the "architectectonic design unit" of airframe construction. A skin panel usually spans lengthwise several frames / rings / bulkheads and spans azimuthally several stringers / longerons / stiffeners. A skin bay refers to the unit contained by two stringers and two frames.

$$^{j}_{k}P_{design} \;=\; ^{j}_{k}P_{in} \;+\; ^{j}_{k}P_{eff} \;=\; (\,^{j}_{k}\sigma_{cr} * \,^{j}_{k}A_{i}\,) \;+\; (\,^{j}_{k}\sigma_{b} * \,^{j}_{k}A_{e}\,) \;=\; ^{j}_{k}\sigma_{design} * \; b \; * \; ^{j-1}_{k}t$$

The load in the effective area, $^{j}_{k}P_{eff} = \,^{j}_{k}A_{e} * \sigma_{b}$, is not limited. The load in the ineffective area $^{j-1}_{k}P_{ineff} = \,^{j}_{k}A_{i} * \sigma_{cr}$, is limited by a "maximum" compressive stress of σ_{cr}.

$$^{j-1}_{k}P_{i} \;=\; ^{j}_{k}A_{i} * \sigma_{cr} \;=\; (b - w_{ave})\, t \;\sigma_{cr} \;=\; bt\,\sigma_{cr} \;-\; w_{ave}\, t\, (\,\sigma_{cr}^{\,2}/\,\sigma_{b})$$

$$^{j-1}_{k}P_{e} \;=\; ^{j}_{k}A_{e} * \sigma_{b} \;=\; w_{ave}\, t\, (\,\sigma_{cr}/\,\sigma_{b})\,\sigma_{b} \;=\; w_{ave}\, t\, \sigma_{cr}$$

So the effective load is always available without correction. Whereas the ineffective load requires calculations which include σ_{b} *and* correction of w_{ave} ($^{j}_{k}A_{e} = w_{ave}\, t$). The new total load comes from these relations. This makes the new thickness a quadratic function of the old ^{j-1}t.

$$
\begin{aligned}
^{j}_{k}P_{design} \;&=\; ^{j-1}_{k}P_{e} + \,^{j-1}_{k}P_{i} \;=\; w_{ave}\,^{j-1}t\,\sigma_{cr} \;+\; b\,^{j-1}t\,\sigma_{cr} \;-\; w_{ave}\,^{j-1}t\,(\,\sigma_{cr}^{\,2}/\,\sigma_{b}) \\
&=\; ^{j-1}_{k}t\,\sigma_{k\,cr}\,\{\, b \;+\; (1 - \,^{j}_{k}w_{ave}\, k\,)\,(\,\sigma_{cr}/\,\sigma_{b})\,\} \\
&=\; ^{j-1}_{k}t\,\sigma_{k\,cr}\,\{\, b \;+\; ^{j}_{k}w_{ave} \;-\; ^{j}_{k}w_{ave}\, k\,\sigma_{cr}(\,^{j-1}_{k}A_{total} / \,^{j-1}_{k}P_{total})\,\}
\end{aligned}
$$

where $\qquad ^{j}_{k}w_{ave} \;=\; 1.7(\,^{j-1}_{k}t)\,\mathrm{sqrt}(E/\sigma_{b}) \;=\; 1.7\,(\,^{j-1}_{k}t\,)\,\mathrm{sqrt}(\,^{j}_{k}A_{total}\, E \,/\, ^{j-1}_{k}P'_{total}\,)$

so $\qquad ^{j}_{k}t \;=\; ^{j-1}_{k}t\,\sigma_{k\,cr}\,\{\, b \;+\; ^{j}_{k}w_{ave} \;-\; ^{j}_{k}w_{ave}\, k\,\sigma_{cr}(\,^{j-1}_{k}A_{total} / \,^{j-1}_{k}P_{total})\,\} \,/\, (\, b\,\sigma^{j}_{k\,design})$

But we do not need to calculate the new thickness[35]. SOL 200 takes care of the calculation of the design stress, σ_{design}, from the analysis stress, $\sigma_{analysis}$, in order to change the thickness as if we had used the equation shown above.

The new thickness, $^{j}_{k}t$, should give us this new load, $^{j}_{k}P_{total} = \,^{j-1}_{k}P_{e} + \,^{j-1}_{k}P_{i}$, but it does not. The FEA results give us a new load according to the new thickness (thinner gives more load) and how much less load this element draws due to the reduced stiffness (thinner increases the load drawn). If we estimate membrane thickness to be approximately proportional to the thickness, we can scale $^{j}_{k}t$ up proportionately to draw in the required load. In the manual method this difference is due to our hand calculations keeping A_{ineff} in the elasto-plastic regime while the rest of the section draws a proportional load. SOL 200 does not give us this distinction[36] directly. So we have to create it.

If we call k' the stiffness that would draw in $^{j}_{k}P_{total}$ and k the stiffness given by $^{j}_{k}t$, the internal load modification ratio for SOL200 optimization iteration input is k'/k (to modify the new thickness giving, $^{j}_{k}t'$).

$$
\begin{aligned}
^{j}_{k}t' \quad &=\quad ^{j-1}_{k}P_{design} \,/\, (\, b\,\sigma^{j-1}_{k\,b}) \;=\; (\,^{j}_{k}\sigma_{cr} * \,^{j}_{k}A_{i}\,) \;+\; (\,^{j}_{k}\sigma_{b} * \,^{j}_{k}A_{e}\,) \,/\, (\, b\,\sigma^{j-1}_{k\,b}) \\
&=\quad [\,(\,\sigma_{cr}/\,\sigma_{b})\,\{\,(\,b - w_{ave}\,)\, t\,\}\,/\, b\,] \quad + \quad [\,w_{ave}\, t \,/\, b\,]
\end{aligned}
$$

$$
\begin{aligned}
(\,^{j}_{k}t' \,/\, ^{j}_{k}t\,) \;&=\; (k'/k) \;=\; (\,\sigma_{cr}/\,\sigma_{b}) \;+\; [\,w_{ave}\, t \,/\, b\,]\,(1 - \sigma_{cr}/\,\sigma_{b}) \\
&=\; (\,\sigma_{cr}/\,\sigma_{b}) \;+\; (\, t \,/\, b\,)\,(1 - \sigma_{cr}/\,\sigma_{b})\,[1.7\,\mathrm{sqrt}\,(\, E/\,\sigma_{b})
\end{aligned}
$$

Notice that the ratio $(\,^{j}_{k}t' \,/\, ^{j}_{k}t'\,) = (k'/k)$ is *not* a direct iteration formula for calculating the new neutral axis. This is a correction for changes in FE model stiffness based on changes in CQUAD4 thicknesses.

---

[35]   Forcing a formula for ^{j}t could be another way to do opti-buckling for the task of calculating the elasto-plastic effects.
[36]   Not unless we use nonlinear material *and* two coincident quads, an effective and an ineff. quad, per location, which is counterproductive.

The lesser stiffness k' would give a smaller load , $_k^jP'_{total}$. opti-buckling can use $_k^jt'$ to approximate this stiffness correction. This is correction requires a DEQATN for the DRESP that is quadratic in the uncorrected thickness. Ideally we could use the correction feature in SOL 200. MSC/NASTRAN provides APRCOD = 3 on the DOPTPRM entry for inverse relationships with the DRESP design objective (MSC/NASTRAN stress output). This is an opportunity for improvement of the method, to find a way to get have MSC/NASTRAN disable automatic removal of APRCOD = 3. Current lower limits for thickness in our models are 0.001 but the program appears to fear that it will divide by a small number.

The use of $_k^j{}_{ave}$ rather than $_k^j{}_b$ indicates that CQUAD4 centroid membrane stress output is equivalent to the membrane stress from vehicle bending.
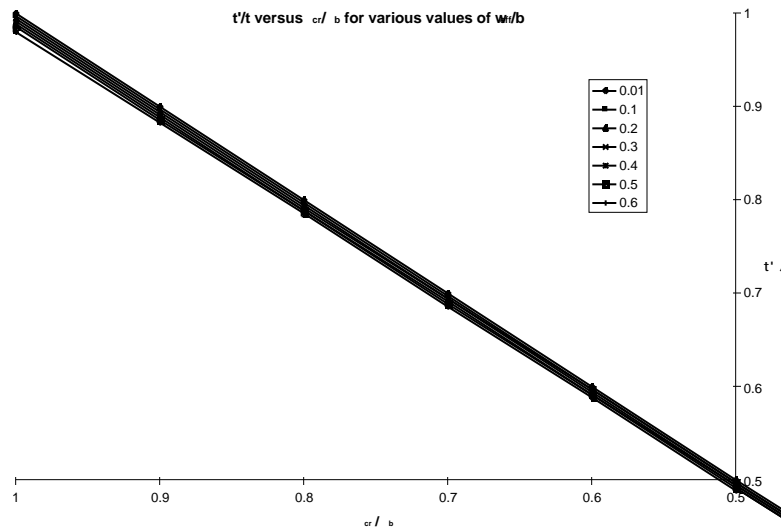

Figure_2:   t'/t correction is nearly linear where it counts

Figure 2 shows that the stiffness reduction does not have nonlinear behavior in the region where we want to use it. The lines get closer for smaller thicknesses. So for an automated version we can optimize with t' shown above using some DMAP for a custom DCONSTR.

To make the entry of these values fully automatic we need the curvcritstress program to calculate $_kb$ and $_k{}_{cr}$. Each SOL 101 run within SOL 200 gives us $_k^j{}_{ave}$. We get E from the material entry for element k in order to calculate w. The analyst provides the curvature and skin panel center of gravity (arm from the geometric center) and $_{cr}$. Especially in the case of p-elements, all the information we need is available.

Optimization Thickness, Four Bar Fiber *Example*:   The opti-buckling method requires a thickness less than that our use of classical Bruhn effective width calculations would give. This comes about due to Bruhn's assignment of $_{cr}$ to the ineffective sheet area (t*b' = A - A$_e$). Bruhn's method, copied in our spreadsheet[37], is not only complete with nonlinear buckling for non-uniform curved sheet (empirical data), it also assigns nonlinear material properties (elasto-plastic). The "ineffective" panel section area carries exactly $_{cr}$. So in the FEA opti-buckling method, if we merely assign t$_{cr}$, the stress from the next iteration

---

37   http://www.absolute.org/bruhn1trial.txt

will be more than $_{cr}$ due to the reduced section not chasing all the load out of that panel (we want to run a linear[38] SOL 200). The following simplified 4 bar fiber example shows how.

This example does not use a combination of webs and stiffeners so we need to buckle the stiffeners themselves. Constraints against motion normal to the orientation of the bars act to make this problem similar to, but different than a stiffened web structure.
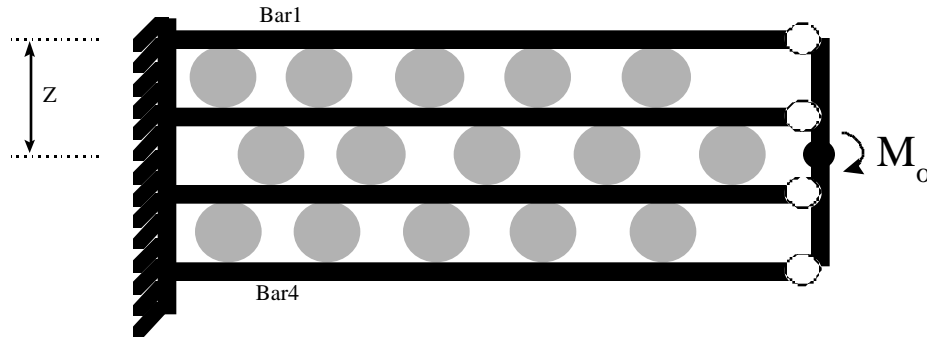

Figure 3:   4 bar fiber model of fibers in a "1-D" version of Bruhn's method,   the reason for $^{j}t < t_{cr}$

If $P_4 > P_{critical}$ (Bar 4 is buckling) the situation is different:
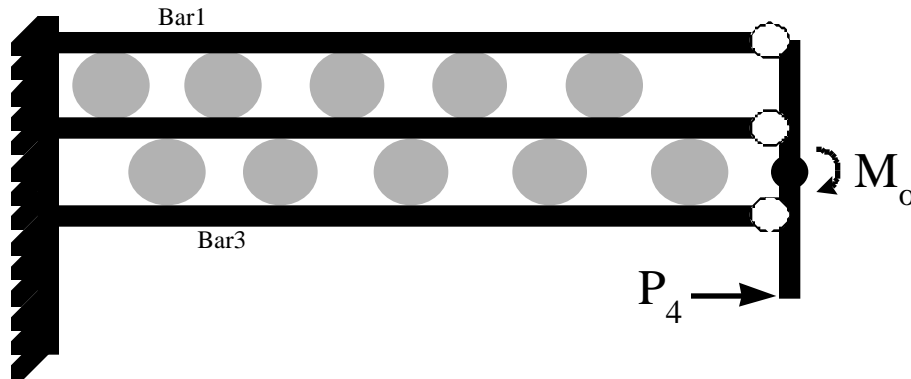

Figure 4:   Four bar model of four fibers with bar 4 only providing critical buckling load reaction

Summing Moments about points 2 and 3 (the "free" end of the "beam" at bar 2 and 3) gives two equations in three unknowns. Unfortunately, the moment summation about point 1 (the "free" of bar 1) and the sum of forces are linearly dependant. However, if we assume that these bars act as a beam (cautions voiced later) we can assume a linear relationship between the reactions, $P_3 - P_2 = P_2 - P_1$, giving the last solution matrixrow. Assume an applied moment of $-M_o$. MSC/NASTRAN results validate this assumption exactly.

$_{pt.2}M_i = 0 = -zP_1/3 \quad +0 \quad +zP_3/3 \quad\quad -2zP_4/3 \quad +M_o$

$_{pt.3}M_i = 0 = -2zP_1/3 \quad -zP_2/3 \quad +0 \quad\quad -zP_4/3 \quad +M_o$

$_i F_i \quad = 0 = P_1 \quad -2P_2 \quad P_3$

Which gives us the following matrices to solve.

1   0   -1   2P_4 - 3zM_o

___

[38]   However, for comparison we will also run nonlinear optimization (optimization of SOL 106 runs) for this "4 bar fiber" model.

| 2 | 1 | 0 | | $P_4 \quad - \quad 3zM_o$ |
| 1 | -2 | 1 | | 0 |

Which gives the following solution matrix for bar forces, where $\quad = \quad = \quad EA^2/(4L^2)$.

1	0	0	$2P_4/3 \quad - \quad 3M_o/2z$		$= P_1$
0	1	0	$-P_4/3$	$= P_2$	
0	0	1	$-4P_4/3 \quad + \quad 3M_o/2z$		$= P_3$

So Bar 2 is in tension, assuming $P_3 > P_{cr}$. (We assumed that only $P_4 > P_{cr}$.) However, it is possible for bar 3 to also be in tension, along with bar 4, if $M_o > 8zP_4/9$.

What is the moment, M_{cr4}, when P_4 just equals P_{cr}? Use the fact that at this critical load $P_4 = -P_1$ or $P_3 = -P_2$. This gives us a view of the situation just as bar 4 buckles which we can compare to the situation with M_{cr4} which is just when bar 3 joins the buckled state with bar 4.

For $\quad M_{cr4} = 10zP_{cr}/9 \quad$ only,

$$P_1 \quad = \quad -P_{cr}$$
$$P_2 \quad = \quad -P_{cr} / 3$$
$$P_3 \quad = \quad +P_{cr} / 3$$
$$P_4 \quad = \quad +P_{cr}$$

$M_{cr3} = 14zP_{cr}/9 \quad = \quad 1.40 \ M_{cr4}$

So an increase in load of 40% will bring us into a new structural load organization with half of the longitudinal stiffeners buckled[39].

A bar model of the moment distribution cap (load application distribution "structure") works only if it acts like a rigid body. Vertical bars 4 times the area of the "fibers" connecting the "free" and still free to rotate ends of "fiber" bars gets us a P_4 within 1% of P_{cr}. If the vertical bars are the same size (A = 0.00785 in^2), for this 10 inch long model, the element forces are not the correct sign since the "cap" is rotating and bending at the "free" ends of the inner "fiber" bars. Modeling the cap with RBAR's gives $P_4 = P_{cr} = EA^2/(4L^2) = (1.07e+7)(0.00785in^2)^2/(4[10in]^2) = 5.1786$ pounds exactly.

"opti-buckling" file entries for SOL 200:   The SOL 200 details for the 4 bar fiber example follow the pin-ended rod example in the MSC/NASTRAN *Design Sensitivity and Optimization* example, page 80. The entries in Table 1 below serve to control the FE model of Bruhn's example problem A20.4 to get SOL 200 to act similarly to Bruhn's manual method. This includes several design variables, DESVAR, of a thickness for every row of panels, the 1st (crown skin) being t1, the 2nd is t2, et cetera.

The DVPREL entries define our Design-Variable-to-Property-RELationships. Since the thickness ^j_kt' is the same thickness variable in the PSHELL entry we can use DVPREL1. An example of the need for DEQATN in DVPRELi entries would be variation of a web or flange dimension where the PBAR entry only uses A and $I = bh^3/12$.

The REGION field of the design response entry DRESP1 allows us to screen constraints. Since skin bays between a particular pair of stringers (all at the same bending arm, z) will tend to have bending stress values in approximately the same range, putting them in a group will help remove unnecessary optimization calculations. The str# is our range identification, it is the integer that is the number for the stringer located above the panel. The MSC/NASTRAN Quick Reference Quide (QRG) Appendix A gives the stress item codes for normal stress in the local element x and y directions (3 and 4) and for major principle and minor (7 and 8) for fiber Z1. We set fiber Z1 to zero on the PHELL entry's second line, first field, in order to measure only the membrane stress associated with bending of the stiffened shell structure.

---

[39]   This 4 bar fiber example does not use a combination of webs and stiffeners. So we need to buckle the stiffeners themselves. Our constraints against motion normal to the orientation of the bars act to make this problem similar to, but different than a stiffened web structure.

The opti-buckling method does not need sensitivities to change the thickness since we know how much to change the thickness based on the stress response. SOL 200 gives the essential objective vector direction. For now, calculation of a design stress as a function of the analysis stress causes MSC/NASTRAN to automatically calculate the proper design space change vector magnitude. This is the kernel of the method, to use the manual Bruhn method to calculate the optimization. This paper does not discuss design sensitivities. The Design OPTimization PaRaMeters entry, DOPTPRM considers the design objective as shown in the t' calculation above. We use a convergence criterion of 5% but we only need to get there once. The manual iteration of SOL 101 runs is the proof of convergence for this opti-buckling method.

We know how far the optimization is going to go, so set DELP to 0.95 to allow the optimizer to change design variables by up to 95% per iteration (the default is 0.2). Use an equality constraint or close to it so that the stress in DCONST is nearly equal for both the lower bound, l_allow, and the upper bound, u_allow. l_allow = $_{k\ cr}$ - , u_allow = $_{k\ cr}$ + (coded in NASTRAN as crit1 +/-).

<u>Table 1</u>   <u>Optimization Entries for A20.4</u>

```
$ Case Control
DESOBJ(max)    =       2010     $ It would be nice to find a way to have more than one objective
DESGLB         =       299
$
$ Bulk Data
$     DSVRid  name   to    lwr_bnd upr_bnd Delxv
$ESVAR 1110   t1     0.032  0.005  0.032
DESVAR 1110   t0     0.032  0.005  0.032
DESVAR 1111   t1     0.032  0.005  0.032
DESVAR 1112   t2     0.032  0.005  0.032
DESVAR 1113   t3     0.032  0.005  0.032
DESVAR 1114   t4     0.032  0.005  0.032
DESVAR 1115   t5     0.032  0.005  0.032
DESVAR 1116   t6     0.032  0.005  0.032
$
$ Independant Design Variable Basis Vector Components
desvar  1010    a    1.0  -1.00  1.000
desvar  1012    b    0.0  -1.00  1.000
desvar  1014    c    0.0  -1.00  1.000
$        DVPid   type  PID   FID   Pmin  Pmax  C0
$dlink    id depdvid  cords   mult indpid1  coef1 indpid2  coef2+
$+    indpid3  coef3
dlink   1210   1110          0.032  1010  1.0   1012  0.000+dlink00
+dlink00  1014    0.0
dlink   1211   1111          0.032  1010  1.0   1012  0.005+dlink01
+dlink01  1014   0.167
dlink   1212   1112          0.032  1010  1.0   1012  0.037+dlink02
+dlink02  1014   0.333
dlink   1213   1113          0.032  1010  1.0   1012  0.125+dlink03
+dlink03  1014   0.500
dlink   1214   1114          0.032  1010  1.0   1012  0.297+dlink04
+dlink04  1014   0.667
dlink   1215   1115          0.032  1010  1.0   1012  0.579+dlink05
+dlink05  1014   0.833
dlink   1216   1116          0.032  1010  1.0   1012  1.000+dlink06
+dlink06  1014   1.0
DVPREL1  5110  pshell    10    4  0.005  0.032           +dvprela
+dvprela  1110   1.0
DVPREL1  5111  pshell    11    4  0.005  0.032           +dvprelb
+dvprelb  1111   1.0
DVPREL1  5112  pshell    12    4  0.005  0.032           +dvprelc
```

```
+dvprelc  1112   1.0
DVPREL1   5113 pshell    13   4  0.005  0.032              +dvpreld
+dvpreld  1113   1.0
DVPREL1   5114 pshell    14   4  0.005  0.032              +dvprele
+dvprele  1114   1.0
DVPREL1   5115 pshell    15   4  0.005  0.032              +dvprelf
+dvprelf  1115   1.0
DVPREL1   5116 pshell    16   4  0.005  0.032              +dvprelg
+dvprelg  1116   1.0
$ DRESP1    2010majorpr0  stress   elem          3        10+dresp1a
$ +dresp1a    11    12    13    14    15
$ possible bug , item 7, CQUAD4 Major Priciple, gives fatal on rods, even if rods are not selected.
$ 3 is x-stress for CQUAD4.
DRESP1     2010majorpr0  stress   elem        3         10
DRESP1     2011majorpr1  stress   elem        3         11
DRESP1     2012majorpr2  stress   elem        3         12
DRESP1     2013majorpr3  stress   elem        3         13
DRESP1     2014majorpr4  stress   elem        3         14
DRESP1     2015majorpr5  stress   elem        3         15
DRESP1     2016majorpr6  stress   elem        3         16
$ DSCREEN stress
$ It is probably OK to use DSCREEN defaults of -0.5 and 20 constraints retained.
DOPTPRM aprcod 1      delp  0.9    iscal  1     itmax  5     +godo
+godo itrmop 1     ptol  2.0
$ these stress targets are the results of the fourth manual iteration fo2m3f06.txt
dconadd   299   300   301   302   303   304   305   306
DCONSTR    300   2010-3.4e+04-3.4e+04
DCONSTR    301   2011-3.0e+04-3.0e+04
DCONSTR    302   2012-2.5e+04-2.5e+04
DCONSTR    303   2013-2.0e+04-2.0e+04
DCONSTR    304   2014-1.3e+04-1.3e+04
DCONSTR    305   2015-6.9e+03-6.9e+03
DCONSTR    306   2016-6.7e+02-6.7e+02
$ These are "equality constraints"
```

DEQATN equations can account for the change in stiffness in the approximation phase by modifying our analysis stress into a design stress based on the equations shown above. An alternative method is to modify the load per iteration step or continue iterations of design load calculation. The manual iteration of SOL 101 runs has this design stress calculation in a spreadsheet[40].

Accuracy in Shear Stress: The manual methods developed and taught by Bruhn, Peery, Niu and others involve a lumped area model assumption. Looking at just these "P's and A's" (lumped loads and lumped areas) does not account for variation of shear stress over the section: the   = VQ/(I b) calculation becomes a shear flow and lumped area calculation $q = (V/I) \sum z_i A_i$. Authors such as Bruhn show that for fuselage stress analysis the proper use of these manual methods incurs error[41] of less than 10%. For a vertical shear load, this inaccuracy comes from lumping in the laterally located skin area with the stringers, and lumping vertically aligned areas. This does not account for how shear loading in the center of the web transfers through joints. If we use at least 4 quadrilateral elements per panel in an FE model, we get the increase in shear distribution accuracy and precision for "free" by use of this opti-buckling procedure.

Optimization Iteration Accuracy: Although we should not have to use it, the reduction of membrane force to panel thickness sensitivity changes stops the SOL 200 ("optimization") run in NASTRAN. A pure linear opti-buckling method has an error in stiffness distribution due to under-compensation of section

---

[40]   www.absolute.org/amuc98/desoptest10.csv
[41]   Bruhn, reference 1 or 4, Page A20.6, " . . . If a third trial were used, . . . differences would be quite small. . ." See the "Analysis" section.

area[42].   Opti-buckling takes care of this with the DEQATN corrections (which uses reciprocal approximation). Or you can use of more iterations. Better yet, use the other two methods presented in that "opti-buckling entries for SOL 200" section. Figure 2 shows the error is quite small for all but severely buckled panels.

Accuracy in Modeling: There is an error in curvature calculation due to averaging. This error is the same as the error in calculation of the moment arm to the center of a flat element representing the curved sheet.   Other opti-buckling error versus the methods shown in Bruhn include curvature error from combining panels of different curvature (e.g. in Bruhn's A20.4, bay #5).

In automating the system we introduce the manual errors due to averaging. These include the difference between the bending stress arm, z, used as the average between the panels two stringer arms, the difference between upper and lower effective widths (square roots of the same error), and the use of membrane stress for area Bruhn distributes into upper and lower stringers of different bending moment arm. These errors are all related and discussed below as direct arm error (trigonometric) and effective width error (square root of the stress – arm).

The direct arm error due to using element centroid stress, using the average bending stress arm for the loading to calculate the A_e, *can be a trigonometric relation*. Using θ_1 as the angle up to the lower stringer, θ_3 as the angle up to the upper stinger, and $\theta_3 = (\theta_2 + \theta_1)/2$. The arm to the mid-chord (average between stinger heights) is r (sin[2 θ_2 - θ_1] + sin θ_1). The error is   { r sin θ_2 - r (sin[2 θ_2 - θ_1] + sin θ_1) } / r sin θ_2 .

$$\text{Error}(z_{ave}) = 1 - [(\sin \theta_3 - \sin \theta_1) / \sin \theta_2] = 1 - [\sin \theta_1 - 2 \sin(\theta_1 + \theta_2) \cos(\theta_1 + \theta_2)] / \sin \theta_2$$

Alternatively, the effective width error due to using element centroid stress, using the average arm for calculating the w's, *can come out directly* from the difference of the bending stress divided by the element stress (average). This is a measure of how far off the average of the stresses is from the average stress (approximately the same as vice versa until we get to high error values). After canceling sqrt(M/I) we get the following.

$$\text{Error}(\sigma_b) = \{1/\sqrt{z_+} + 1/\sqrt{z_-} - 2/\sqrt{([z_+ + z_-]/2)} \} / \{2 / \sqrt{([z_+ + z_-]/2)} \}$$

$$= \{ \sqrt{1 + z_+/z_-} + \sqrt{1 + z_-/z_+} \} / (2\sqrt{2})$$

For a 10% difference between z_+ and z_- this gives a ratio of 1.00085 or almost 0.1 % error. z_+ has to be less than half z_- for the error to approach 5%. The error becomes significant (5%) when the difference exceeds 100% (near the lightly loaded neutral axis, but z_{ave} is more accurate and the stress is small there, counteracting this concern).

mid-arc

mid-element

z_+

z_-

curvature calculation

1   2   3
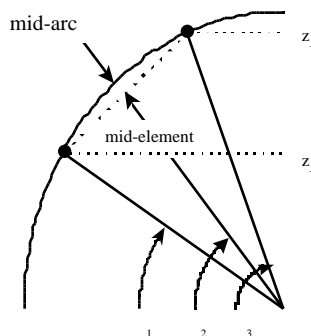
Figure 5:   Curvature calculation for ccs_geom (Curve Critical Stress extraction program).

---

[42]   This is the stiffness issue discussed earlier. Using Bruhn's section area the FEA will draw too much load. Bruhn assumes elasto-plastic material properties for the ineffective area but a linear static SOL 101 solution will assume linear properties and make the member more stiff than Bruhn intended for manual calculations. This is a reason for DEQATN corrections in the approximation phase of Sol 200.

Assumptions: The opti-buckling method includes the following assumptions in Table 2.   A comparison of these limitations to other methods follows in Table 3.

Table 2:   Opti-Buckling Method, Assumptions

Assumptions Shared with all linear methods including the Manual Methods
- Small Deflection
- Linear Material
- Uniform Material Properties
- Omission of stress concentration effects at joints (this is a global *loads model only*[43])

Opti-Buckling Assumptions
- h-mesh calculation of r ➔ $r_{ave} = r_{panel}$
- z_{ave} gives average bending stress.

To get a look at the benefits of using the opti-buckling method consider the error induced from other methods. First, the manual method usually cuts out inclusion of pertinent structural details in order to arrive at a quick result. Additionally, manual shear calculations use plate area lumped into stiffener locations.

Secondly, pure Linear Static FEA with CQUAD4 elements certainly ends up with the wrong neutral axis. Some companies have methods of correction added into the FEA results before or during (e.g. Duberg[44,45,46]) final post-processing. This opti-buckling method is an attempt to standardize, automate, and increase the accuracy of publicly available code.

Finally, Nonlinear Static FEA should give the same results as the opti-buckling method.   Nonlinear buckling can require input of stress – strain curves. The terms of the Green's Strain Tensor (including differential stiffness) do not require this material data, but a fully nonlinear buckling solution should use them. There is a good case for particular vehicles having only been nonlinear in the geometric sense, but you never know for certain. And, almost *all* of these effects are included in the empirical data in Bruhn's book (and others) as nomographs, tables, and charts[47] for curved Aluminum stiffened panels.

Table 3: Comparison: Opti-Buckling Method, Linear Static FEA, Nonlinear FEA, and Manual Calculations

 Manual Method
- Elliptical section
- Local buckling strength calculation (expandable to include pressure and torsion)
- Skin panel thickness variation, stringer area variation
- Other loads require separate calculations and load summation: torsion, pressure, combinations
- Other stress recovery methods established: torsion, pressure, combinations
- Accounts for buckling in skin panels

---

[43]   Caution to FE analysts and stress folks that you label the model a *Loads Only Model* does *not* preclude someone from trying to "push the button" to get "just a peek" of the stresses in the region. Those "unofficial" FEA results often show up with final reports at customers' offices. "Pretty pictures" are irresistible. White out is quite available. Credibility should be a perpetual concern.

[44]   MELCON, M.A., ENSRUD, A.F., Lockheed Aircraft Corporation, "*Analysis of Stiffened Curved Panels Under Shear and Compression*," Journal of the Aeronautical Sciences, February 1953

[45]   DUBERG, JOHN E, "*A Numerical Procedure for the Stress Analysis of Stiffened Shells,*" Journal of the Aeronautical Sciences, August 1949

[46]   Paul Kuhn, James P Peterson, and L Ross Levin, *A summary of Diagonal Tension, Part I – Methods of Analysis*, NACA Technical Note 2661, May 1952

[47]   This data is also available in MIL-HDBK-5G references cited previously and the NASA Astronautics Structures Manual section C.

Linear Static FEA
• Elliptical section
• Kicks, taper, skin panel thickness variation, stringer area variation
• Other loads for "free": torsion, pressure, combinations
• Does not account for buckling in skin panels (possible problems: neutral axis, bottlepin loads, et cetera)

Nonlinear Static FEA
• Elliptical section
• Kicks, taper, skin panel thickness variation, stringer area variation
• Load combinations are not allowed
• Accounts for buckling in skin panels (possible problems: neutral axis, bottlepin loads, et cetera)
• Iterations to Solution is Unknown, prediction of convergence effort ($) for new models is not feasible
• Requires stress – strain curve entry
• Requires Green's Strain Tensor too (including differential stiffness)

Opti-buckling
• Elliptical section
• Local buckling strength calculation (expandable to include pressure and torsion, curvcritpress.c)
• Kicks, taper, skin panel thickness variation, stringer area variation
• FEA globally modifies the fuselage accounting for neutral axis variation along the length of the vehicle
• Other loads for "free": torsion, pressure, combinations
• Expansion to shear from torsion is nearly for "free"
• Accounts for buckling in skin panels
• Stiffness error removed by SOL200, DMAP, or iteration of SOL 101.   Elasto-plastic without nonlinear.

Comparison to Test: Comparison to test for this system is less useful than a comparison to theory. The point is that Bruhn's method is an old standard in the industry and so comparison to these manual methods is the first step for validation of the use of optimization in curved panel stress analysis. Validation of the method to theory does not include many assumptions that validation to test would require. Testing can also contain many inaccuracies and limitations. But there is a more important point. Since our primary objective is to improve on the location of the fuselage FE model neutral axis, we assume that meshes in use already converged for internal, non-buckling loads. The opti-buckling method seeks to improve and extend the effective sheet modifying factors in use by various airframe manufacturers and refurbishers, airlines, non-aerospace stiffened curved shell structures, and as a general method of p-element buckling analysis.

Analysis

Bruhn Results for A20.4 versus

Manual SOL 101 Opti-buckling solution for the A20.4 problem

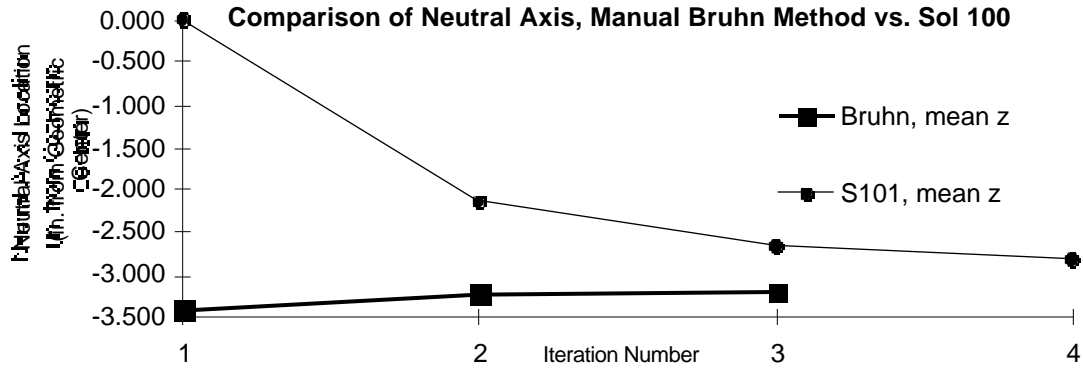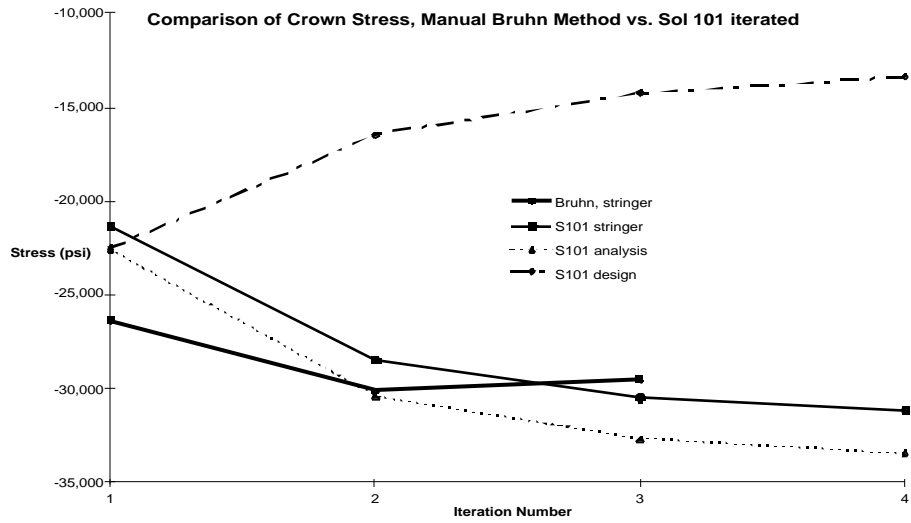| iteration	z_{Bruhn}	z_{S101}	str,Bruhn	str,S101	str,analysis	str,design	$I_{yy,Bruhn}$	$I_{yy,S101}$
1	-3.407	0.000	-26,374	-21,319	-22,556	-22,556	1468	1491.767
2	-3.213	-2.148	-30,087	-28,459	-30,387	-30,387	1430	1251.579
3	-3.193	-2.659	-29,496	-30,449	-32,642	-14,281	1420	1203.796
4		-2.818		-31,138	-33,439	-13,407		1188.807

Figure_6: _____ Neutral Axis Comparison

Figure_7: _____ Stress Comparison

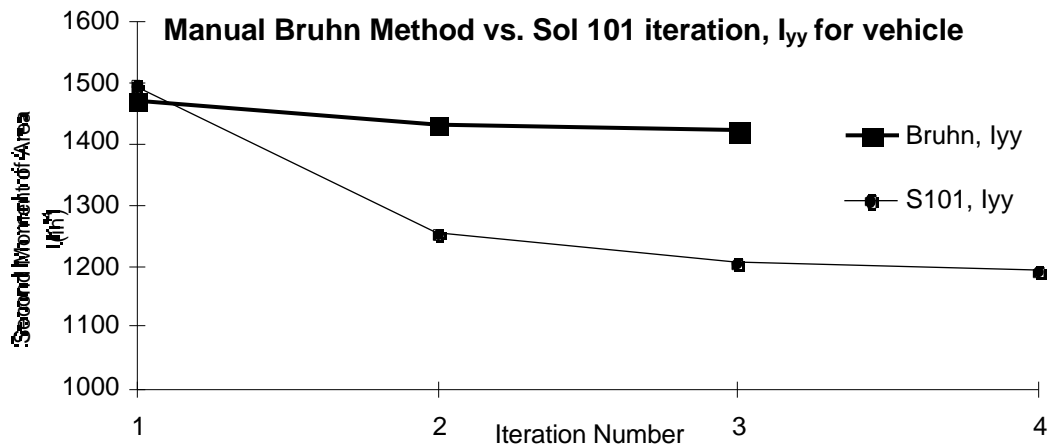Figure_8: _____ Second Moment of Area Comparison

Discussion

Differences in results are due in part to Bruhn's assumption of a linear stress distribution result across the vehicle section from -32 ksi at the top (crown) to +32 ksi at the bottom of this A20.4 example fuselage, and that "plane sections remain plane". FEA shows warping of cross sections. Since each "bulkhead", "ring", or "frame" is made from 24 bar elements with bending and shear stiffness and the skin CQUAD4 elements do not strain uniformly as the manual methods assume the results show warpage. Nevertheless, a plot of Bruhn's A20.4 stringer stress versus moment arm z, compared to a similar plot for the FEA results shows that the manual methods are over conservative for all but the crown, especially at the second stringer set down from the top stringer set. Bruhn's stringer stresses converge more rapidly than FEA because when the FEA assumes no elasto-plastic behaviour the structure is symmetric (or anti-symmetrically loaded in this case). So the first FEA results give mean z = zero. But the FEA iteration quickly reduces this neutral axis location to a correction less than that of Bruhn's. Notice that we performed a third Bruhn iteration[48].

Full Sol 200 optimization runs are proceeding and will be posted when available. Manual Sol 101 iteration convergence is a prerequisite to the use of Sol 200. These manually iterated Sol 101 results converge to a slightly smaller neutral axis offset (12% lower) due to warping of cross – sections, inclusion of frame bending stiffness, and more accurate accounting for load distribution. MSC/NASTRAN results are close to Bruhn's and include warping and other physical effects ignored in hand calculation methods. Plots of load versus moment arm for Bruhn's results versus FEA will be presented at the conference. They show an interesting nature of historical conservatism in hand calculation.

Conclusions

This paper and references should allow for easier implementation of correction to stiffened shell analysis for diagonal tension due to vehicle bending by use of the commercially available MSC/NASTRAN Solution 200 opimization or manual iteration of Sol 101 runs. Reference organization in this paper could help reduce overhead in creation of buckled sheet analysis and development of automation for such analysis. Part of the reasons for creating this method came about while describing airframe stress analysis to my non-aerospace colleagues. There are possibilities for great cost reduction in non-aerospace industries by use of Aerospace handbooks[49] and creation of systems such as Boeing's Stress Reports Systems for each entire vehicle, and by use of methods such as used in this paper. Part of the existing difference between industries is Organizationaly. Customers maintain their own aircraft with their own engineering departments. This is quite different for other industries such as automotive where OEM's effectively do this task in its entirety for their dealers. We expect the aircraft industry to continue having customer engineering departments. Boeing seeks to expand its maintenance business but is unlikely to displace all customer engineering. Changes or modifications to this opti-buckling method can be posted with the rest of this current documentation. Comments and improvements are welcome.

Acknowledgments

---

[i] Key words: Finite Element Analysis (FEA),Finite Element Modeling (FEM), Finite Element model (FE model), optimization, buckling, crippling, effective sheet, fuselage, semi-monocoque

[48] The extension of Bruhn's A20.4 problem for a third iteration, with minor corrections, is available at www.absolute.org/amuc98/effective4s.prn or effective4t.txt.

[49] References such as MIL-HDBK-5G and the NASA Astronautics Structures Manual,

d:\aa\auc98\optibukl\optibuckl17.doc