ESRD, INC.

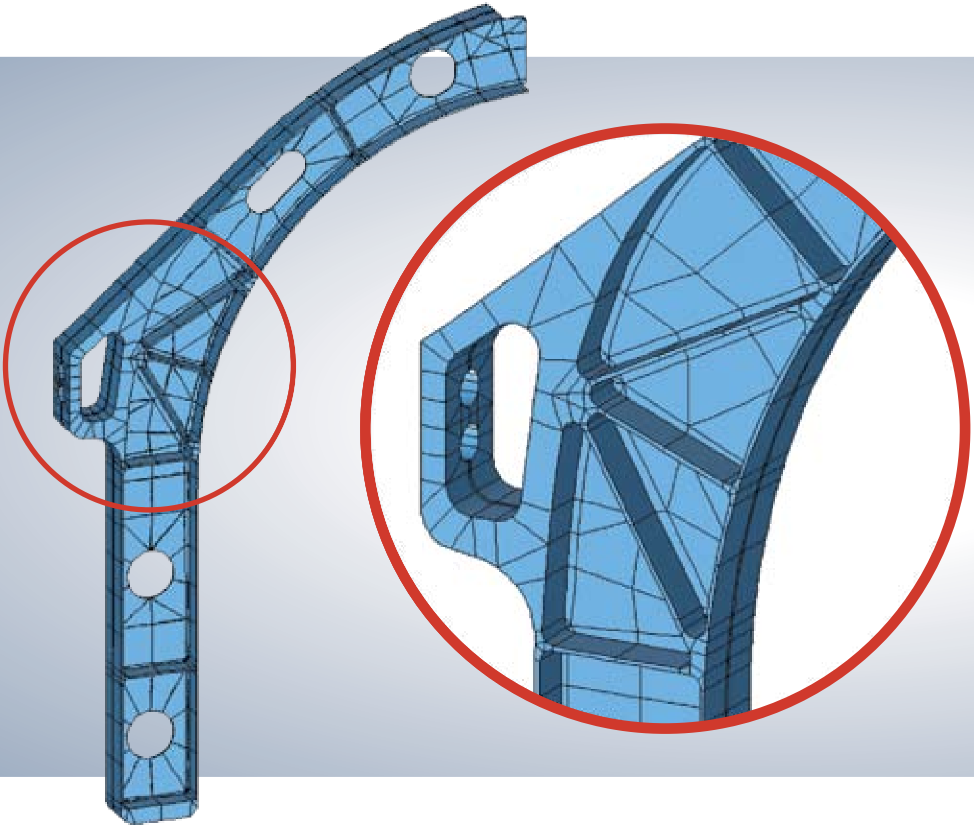
StressCheck®
PRODUCT BRIEF

Buckling Analysis

Eigenvalue (bifurcation) buckling analysis is useful for finding the load factor and corresponding buckling shape for a given set of loads and constraints. Parts are modeled in *StressCheck* using 3D solid or shell elements. These elements can have isotropic, orthotropic, or laminated composite material properties assigned to them.

Many machined parts are optimized for weight and contain thick flanges and thin webs as shown in the example below.

The coexistence of thin and thick domains can present a challenge for traditional h-version FEA codes due to the large number of elements required to properly represent the domain. *StressCheck*, based on the p-version finite element method, has a distinct advantage over the h-version for these types of problems because the buckling mode shapes are approximated well by polynomials of high order. In the case of a very thin domain, multiple layers of h-elements are necessary to capture the bending stiffness which in turn drives the total number of elements in the model to extremely large numbers. By contrast, a single p-element can be used as long as the minimum polynomial order (p-level) is greater than or equal to 4. As shown in the example below, relatively few elements are needed to capture the topology of the part and, most importantly, a single element through the thickness of the web is used. This part was meshed in CATIA using isoparametric hexahedral and pentahedral elements and imported into *StressCheck* via the NASTRAN bulk data input capability.



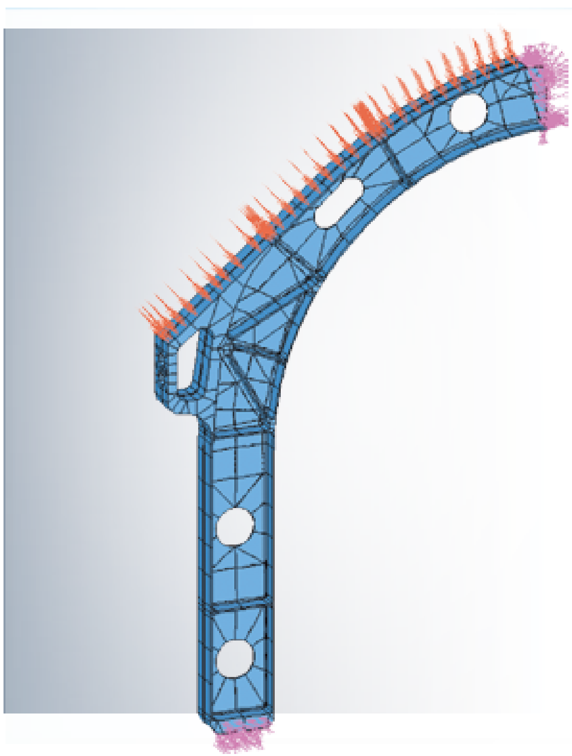
Typical aircraft frame with thin webs and thick flanges

- The p-version has a distinct advantage over the h-version for these types of problems.
- Buckling mode shapes are best approximated by polynomials of high order.
- For very thin domains (e.g. web) only a single element through the thickness is used which dramatically reduces the number of elements in the model.

To perform a buckling analysis, *StressCheck* first solves the linear problem corresponding to the specified loads and constraints. Using the stress field computed from the linear solution, a geometric stiffness matrix is generated and used for eigenvalue computation.

Buckling Analysis

Minor details that add to the complexity of the model, such as small fillets, can be omitted if the information of interest are the buckling load factors and mode shapes and not the stresses. These details add unnecessary degrees-of-freedom (DOF) requiring additional computation time without producing significant changes to the results. In this example the fillets between the flanges and webs were omitted. The ends of the frame are constrained with normal springs representing the stiffness of the connecting parts. A pressure load is applied to the upper-outboard flange.



Upon completion of the analysis, two sets of solutions are available: the linear solution and the eigenvalue buckling solution. There are two post-processing operations that are relevant to buckling analysis: buckling load convergence and display of the buckling mode shapes (shown below).

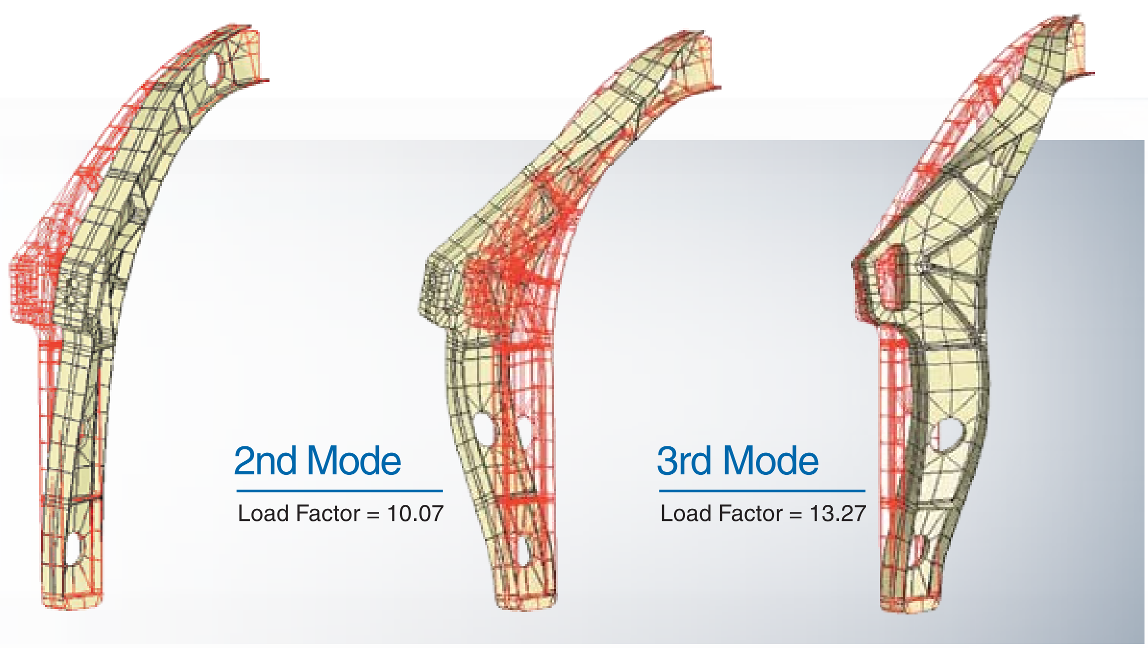
Error Estimate Aircraft Frame
Solution=BUCK_SOL_0001, runs #3 to #6

Run#	DOF	Buckling Load Factor	% Error
3	24162	2.6304	2.35
4	42480	2.5967	1.04
5	69050	2.5856	0.61
6	105786	2.5710	0.38

Estimated Limit 2.6677

Buckling Load Convergence (Mode 1)

Results



1st Mode
Load Factor = 2.57

2nd Mode
Load Factor = 10.07

3rd Mode
Load Factor = 13.27