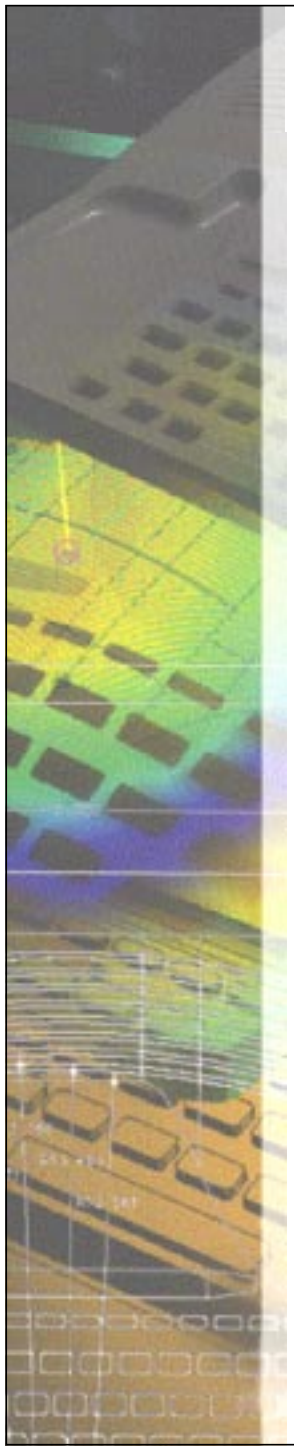


I-DEAS Simulation Advanced Modeling Techniques

Pete Ogilvie
SDRC Products Group
April 1998

SDRC

Introduction

- Purpose
  - An I-DEAS Master Series centric approach to typical finite element modeling situations
- Contents
  - Modeling Bolted Connections
  - Cyclic Symmetry with Interference
  - Modeling Structures with Cables
  - Connecting Beams, Shells to Solids
  - Breakout Modeling
  - Feature Replacement
  - The Amazing Xpanding Rigid Element

Disclaimer


DANGER
USE AT YOUR OWN RISK
IT IS THE
RESPONSIBILITY
OF THE USER
TO CONFIRM THE
ACCURACY AND USEFULNESS
OF THE RESULTS


CAUTION
USE AT YOUR OWN RISK
IT IS THE
RESPONSIBILITY
OF THE USER
TO CONFIRM THE
ACCURACY AND USEFULNESS
OF THE RESULTS


WARNING
USE AT YOUR OWN RISK
IT IS THE
RESPONSIBILITY
OF THE USER
TO CONFIRM THE
ACCURACY AND USEFULNESS
OF THE RESULTS

Modeling Bolted Connections

- Bolted connections have a preload on the bolt, but you can't just apply a force
- Answer, model bolt as a beam element and use temperature load to "shrink" it
- Need to take iterative approach, because preload depends on stiffness of structure between bolt head and nut (which you don't know beforehand)
- Techniques
  - ☛ Model bolt as beam elem, rigid elems from end of beam to washer surface

Modeling Bolted Connections

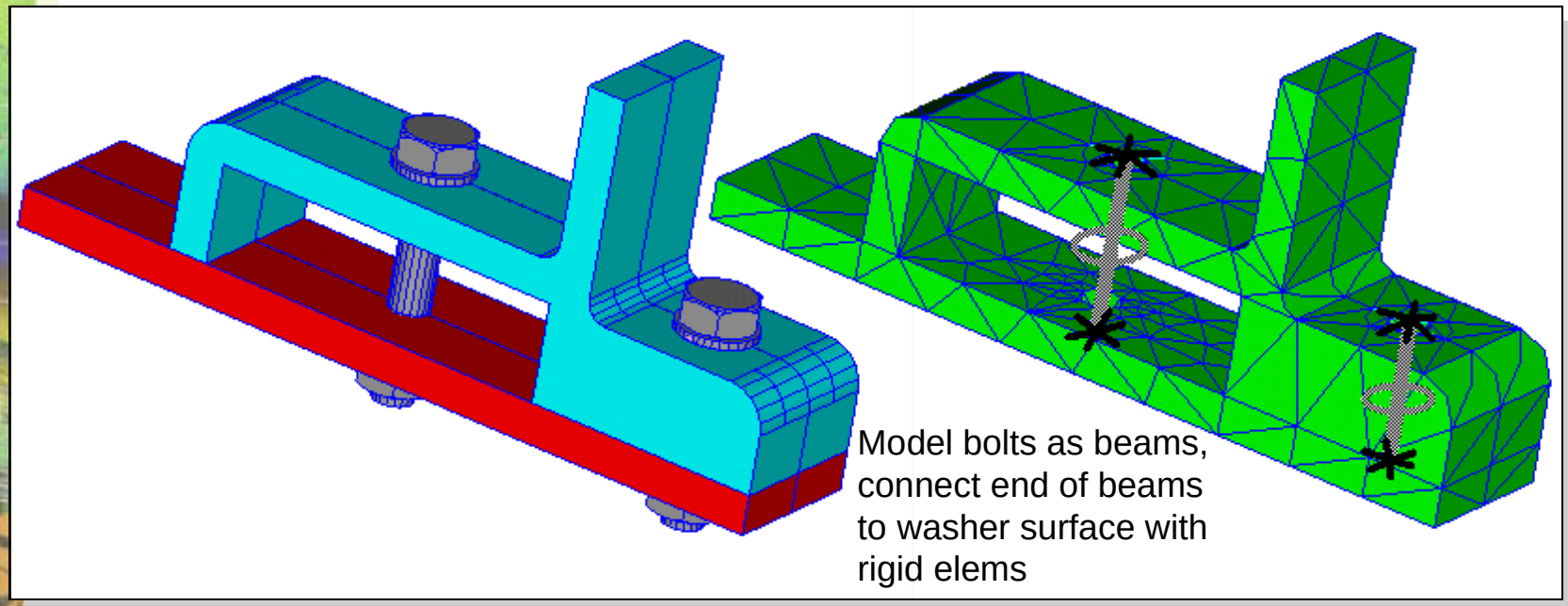
- Determine initial preload force, can be derived from bolt torque recommendations
  - From Shigley, $T=KdF_{pt}$
  - T=Torque
  - d=Nominal Diameter of Bolt
  - F_{pt}=Target Preload
  - K="Bugger Factor", assume .15 for this example
- For a given Coefficient of Thermal Expansion, take a guess at delta T needed (doesn't have to be exact)
  - For Example, if a steel bolt has d=.5 in, L=1.75 in and a CTE of 6.5e-6/DegF, then a delta T of -154 DegF...
    - Shrinks an unrestrained bolt 1.75 * 6.5E-6 * -154 = -.0017 inches, stress free
    - But, if this bolt is joining rigid plates, this would induce an axial stress of 30,000 psi, no displacement

Modeling Bolted Connections

- Now run analysis with these delta Ts on the bolts and look at resultant forces in beams
- Next, for each bolt adjust delta T by factor of Desired Preload/Axial Force and rerun analysis
- Repeat this until you obtain desired accuracy, then add service loading
- Variations on a Theme
  - Use Contact on mating surfaces
  - Use constraints instead of rigids to connect bolt to washer
  - If model has temperature loads, you may have to take different approach
    - Use node to node gaps
    - Use MPCs
    - Nastran DEFORM element
  - Dynamic analysis in Model Solution can use bolt forces to “stress stiffen” a model
  - Use Restart solutions

Modeling Bolted Connections

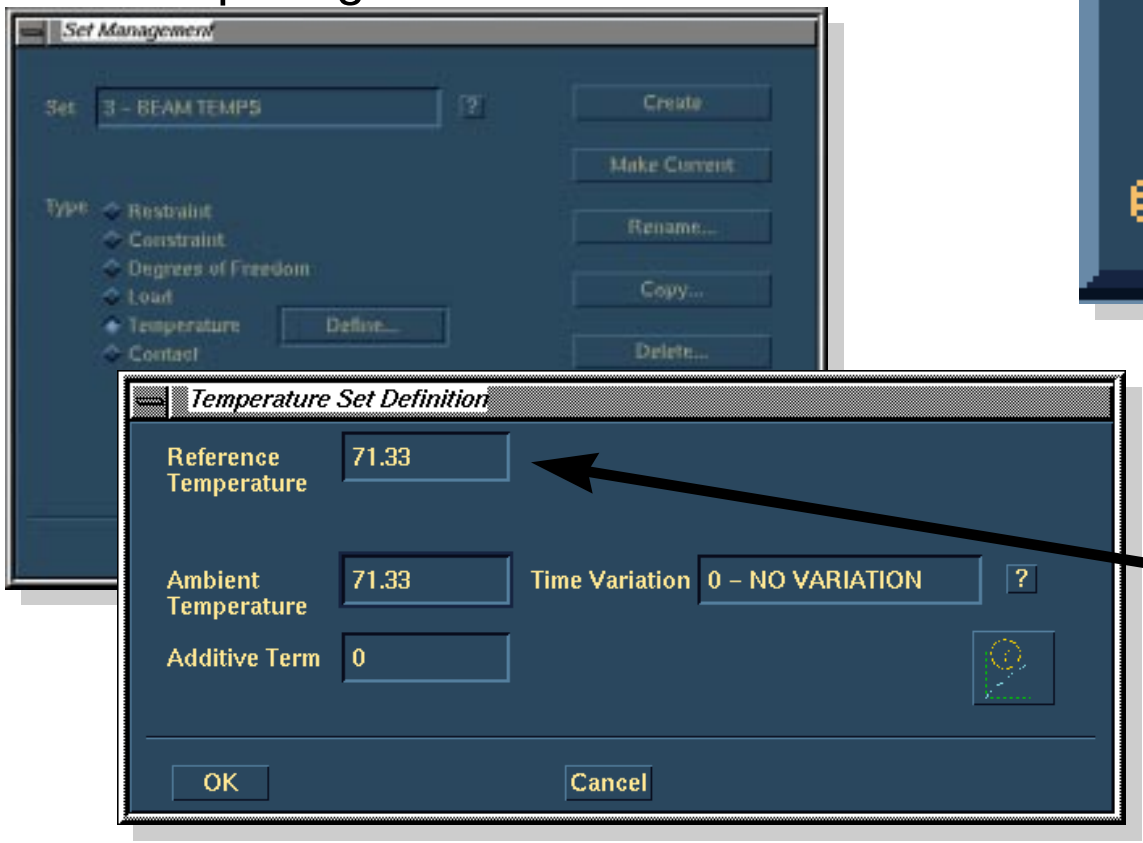
- Example - assembly bolted to 15 foot-pounds, 1/2 in bolts



Model bolts as beams, connect end of beams to washer surface with rigid elems

Modeling Bolted Connections

- Example - guess at initial delta T



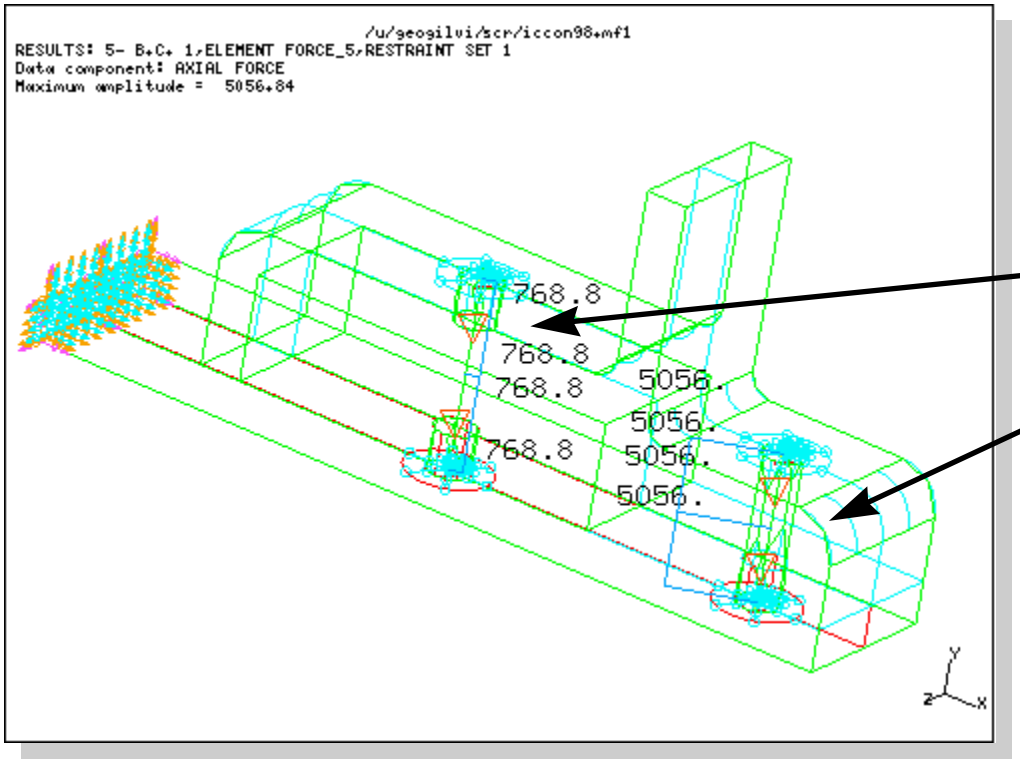
Use Beam Temps to apply delta T

Actual temp used is the delta from this value

For example, if you want to apply a delta T of -154 then the temp on the beam is 71.33-154=-82.67 deg F (for Model Solution solver)

Modeling Bolted Connections

- Example - solve model and look at resulting forces



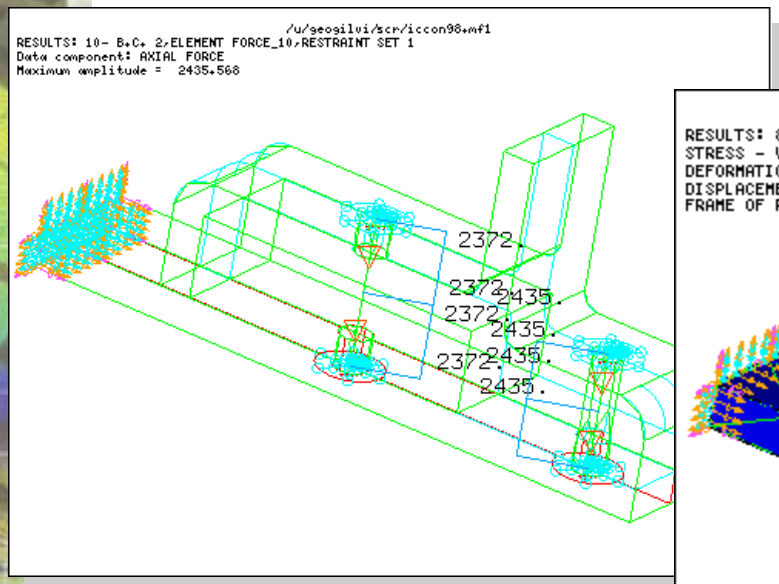
Desired preload is 2400 pounds (derived from $F_{pt} = T/Kd$)

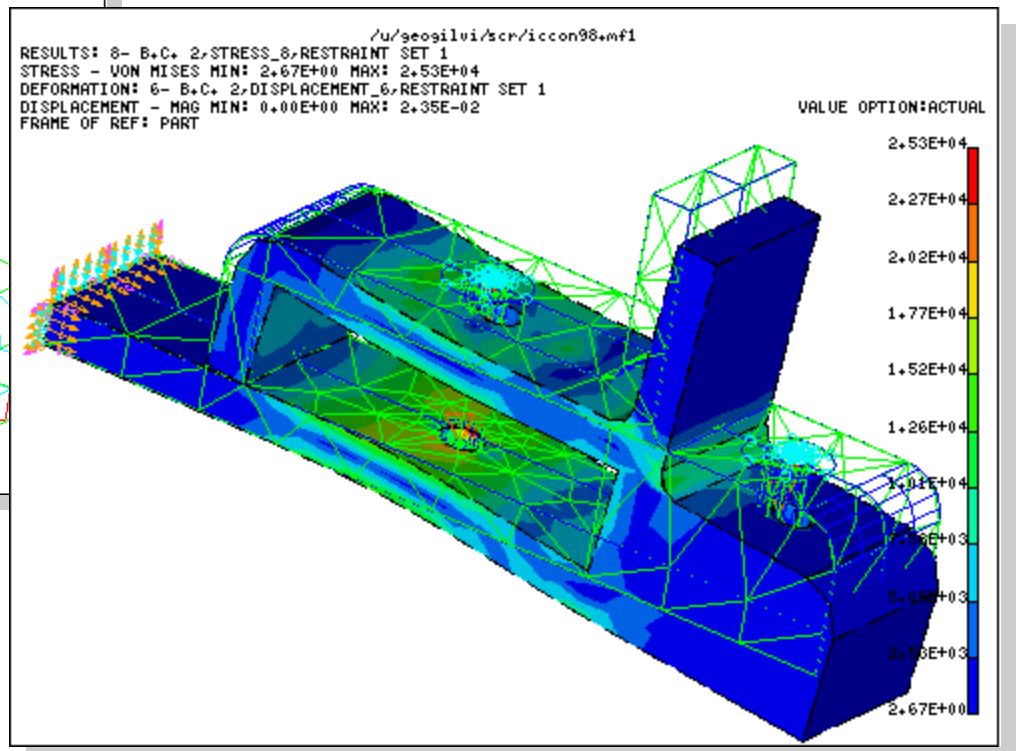
Need to scale this beam delta T by 2400/768.8

and this beam delta T by 2400/5056

Modeling Bolted Connections

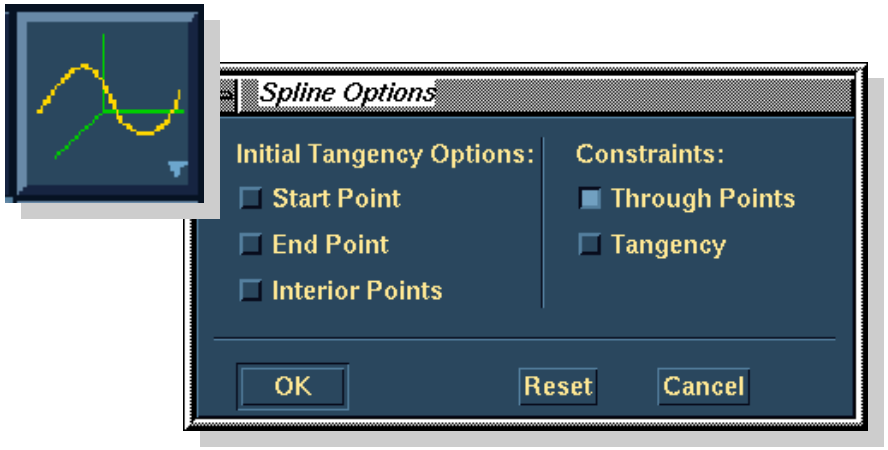
- Example - Solve again and check forces





Modeling Bolted Connections

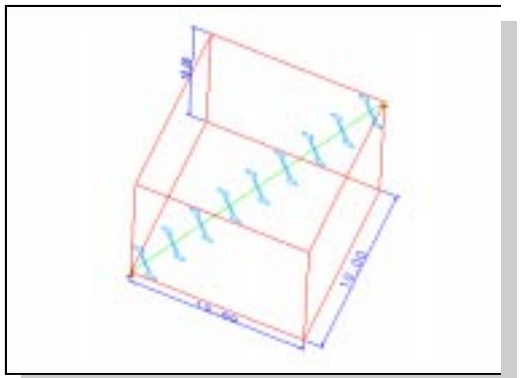
- Tip - A Stealth enhancement made to Master Modeler allows you to build Associative 3D Wireframe

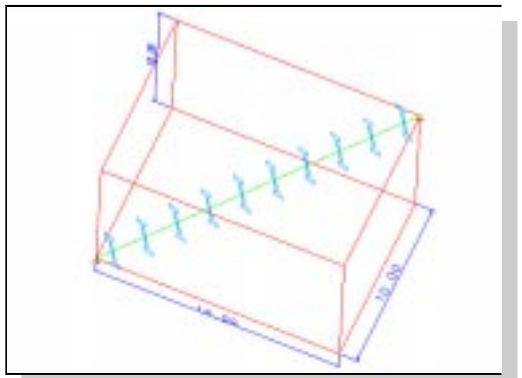


A 3D Spline created between 2 points is actually a line and fair game to generate elements on

Plus, the endpoints will track vertices and reference points!

This will create coincident nodes at the end points, check for this and merge if necessary





Cyclic Symmetry with Interference

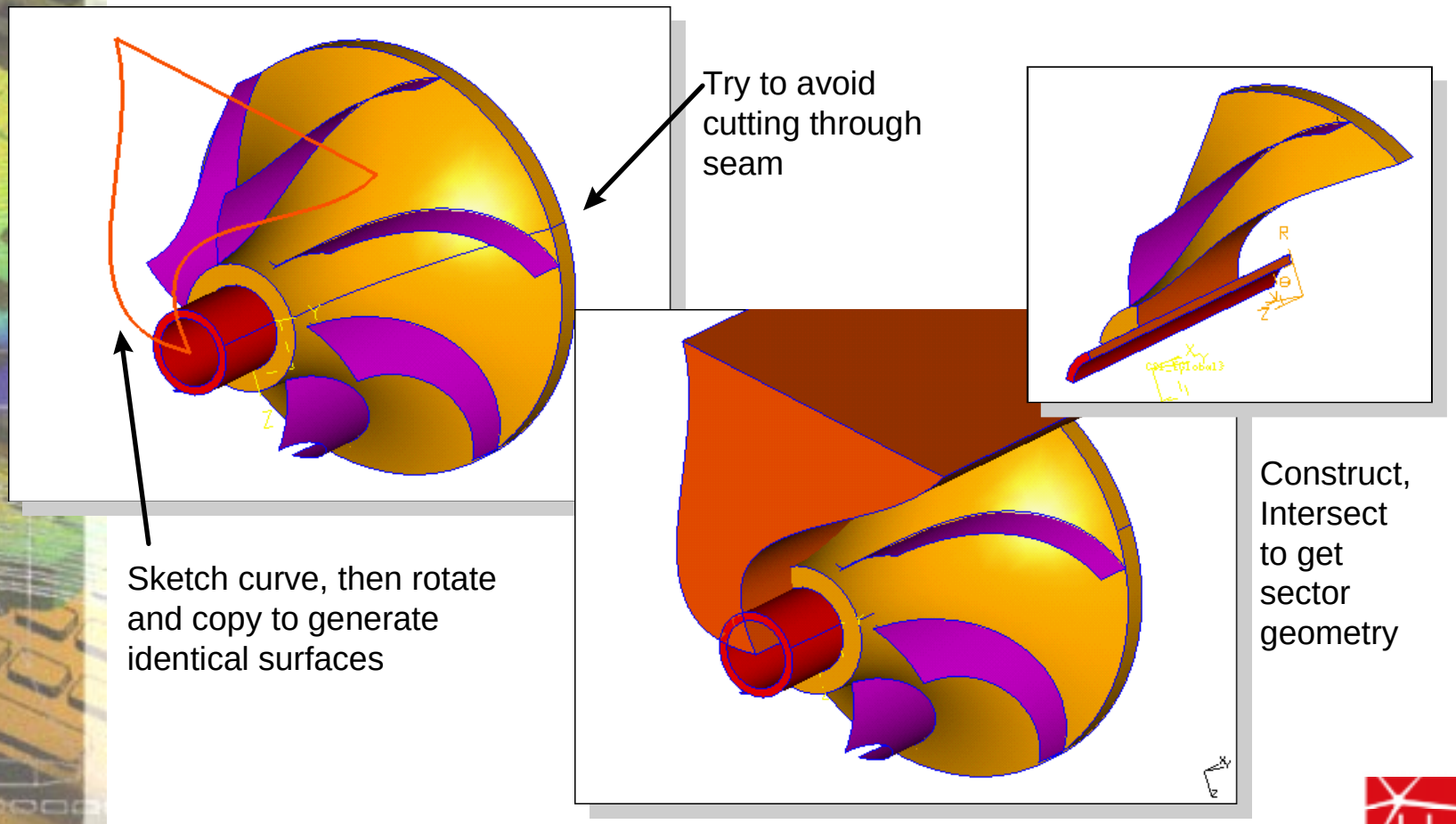
- Modeling sectors of cyclic (or rotational) symmetric parts
- Assumption, loads are…
  - Radial, axial or torque acting equally on each sector of the part
- Technique
  - Slice out sector using identical surfaces as cutters
  - Use surface dependency to force identical meshes
  - Create cylindrical coordinate system
  - Generate mesh
  - Modify node displacement coordinate systems
  - Use couples to connect sides of sector
  - Program file that uses enclosed volume picking can be used to automate

Cyclic Symmetry with Interference

- Model interference using Contact Solution
- Assembly stresses can be significant
- Techique
  - Use contact analysis, model pin and bore
  - Use “Construct, Add” (in Master Modeler menus) to prevent merging of coincident surfaces when creating assembly
  - Can model interference and use negative search distance
  - Can model to nominal and use offset to model initial interference
    - This may be better approach, allows use of surface dependency to get matching meshes

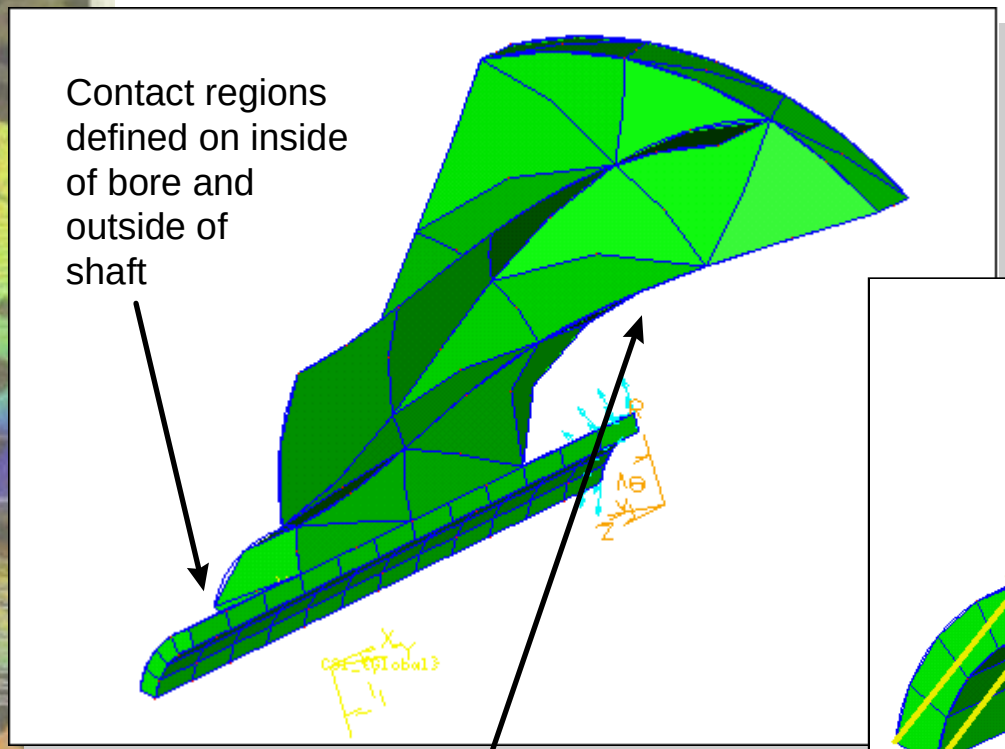
Cyclic Symmetry with Interference

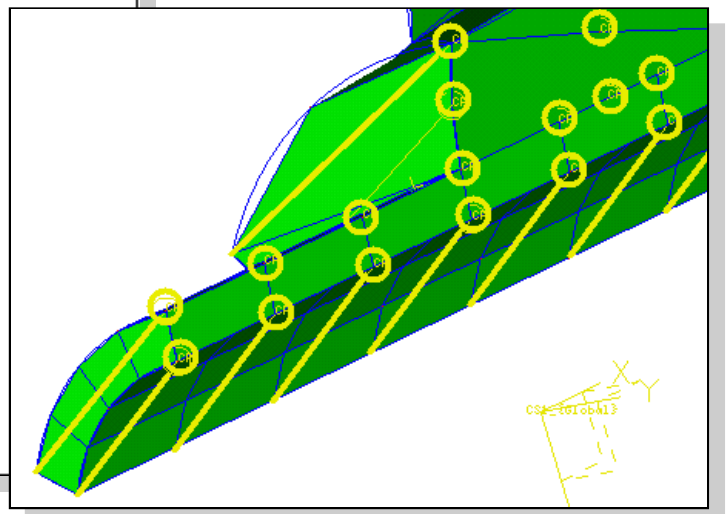
- Example - turbine rotor press fit onto hollow shaft



Cyclic Symmetry with Interference

- Defining Cyclic Symmetry BCs





Use surface dependency to force identical meshes on either side of the slice

Create Coupled DOF from nodes on one side to the other
Couple R, Theta, Z for solids,
R, Theta, Z, RR, RTheta, RZ for shells

Cyclic Symmetry with Interference

- Defining interference fit

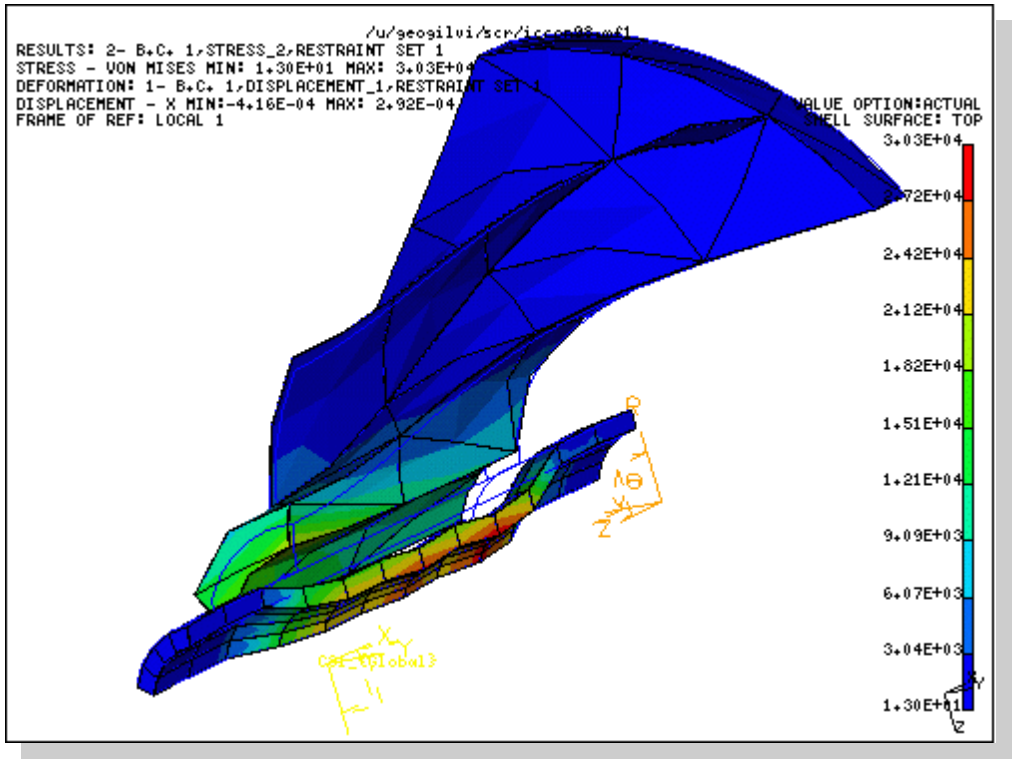
Method 1 - Tube diameter and bore diameter are same

Define interference amount
as a surface offset on one of the contact regions

Method 2 - Model interference on the part, use negative search distance to detect contact pairs

Negative search distance will also search in positive direction

Cyclic Symmetry with Interference



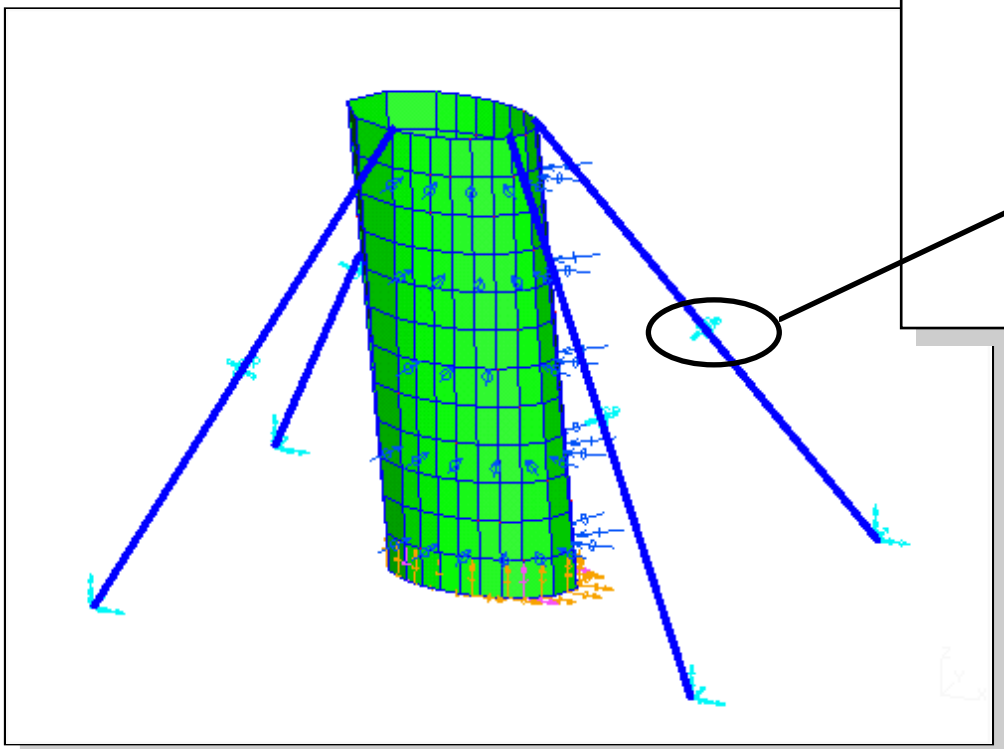
Cyclic Symmetry with Interference

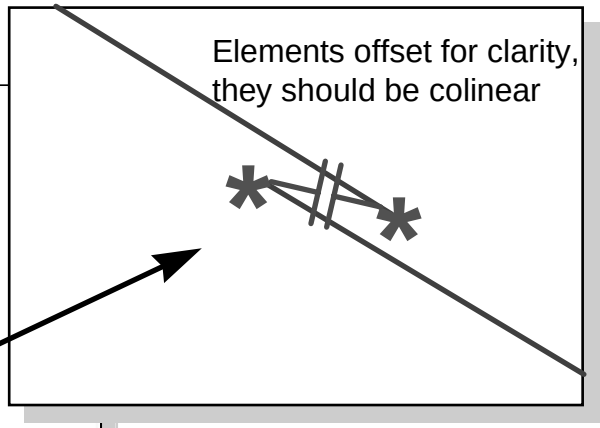
- Tip - Write program file to create couples
  - Align workplane with cylindrical system and change workplane appearance to cylindrical
  - Group nodes on one side of sector, save as list
  - For each node in list
    - List out R, Theta, Z
    - Add Delta Theta to first node to get R, Theta, Z of second node
    - Create couple
      - First node is by label
      - Second node is by RMB “Enclosed Volume”, “Center & Deltas”, “Key In”
        - This will allow the program file to pick the second node by it’s R, Theta, Z position

Modeling Structures with Cables

- Cables take load in tension only
- Technique
  - ☛ Model with beam elements and gaps

Modeling Structures with Cables





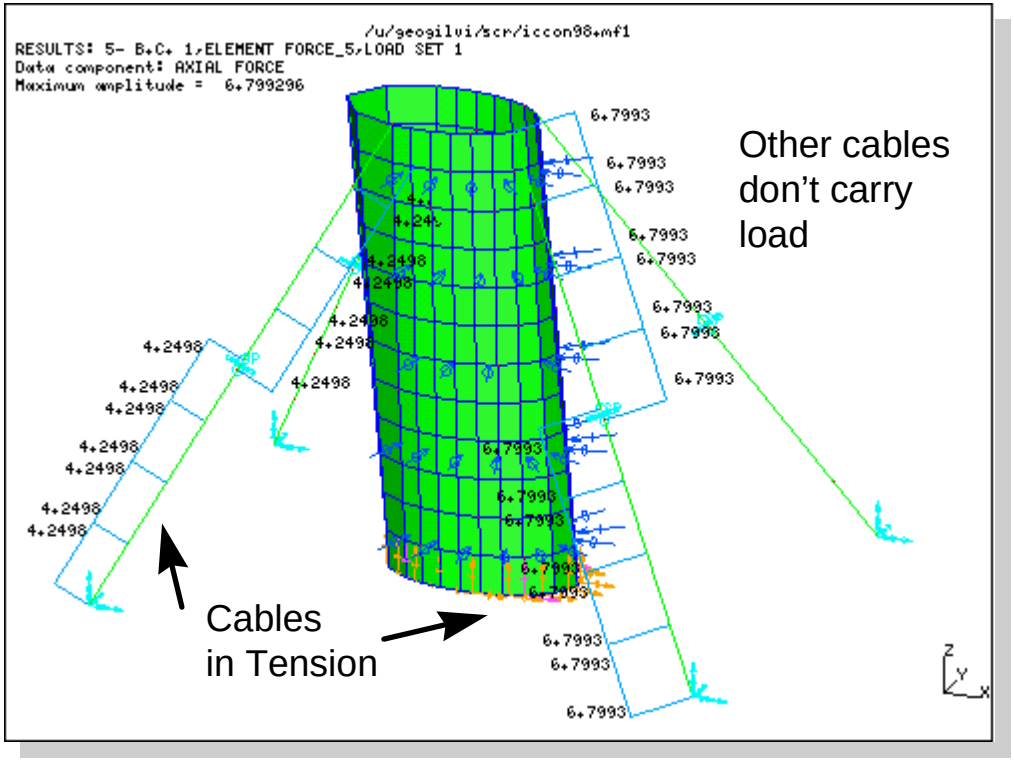
Overlap ends of beams a slight bit, then create uniaxial gap element between them

This creates a compression only gap in Model Solution

A tension on the cable then closes the gap

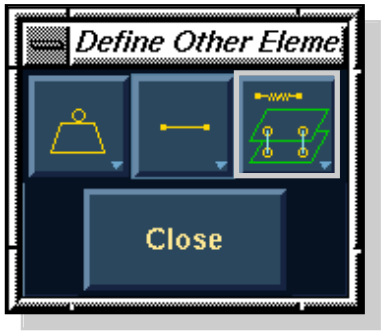
Model cables as beams, you may want to use small Iyy and Izz with clamped bcs to prevent singularities

Modeling Structures with Cables



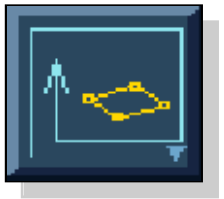
Modeling Structures with Cables

- Tip - Model gaps as geometry based springs, then change family to gaps

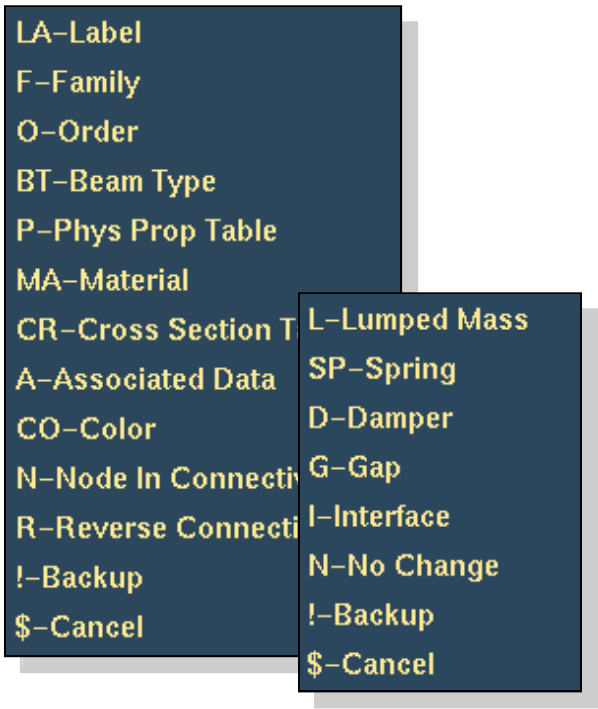


Define gap as point-to-point spring

Generate “other” elem



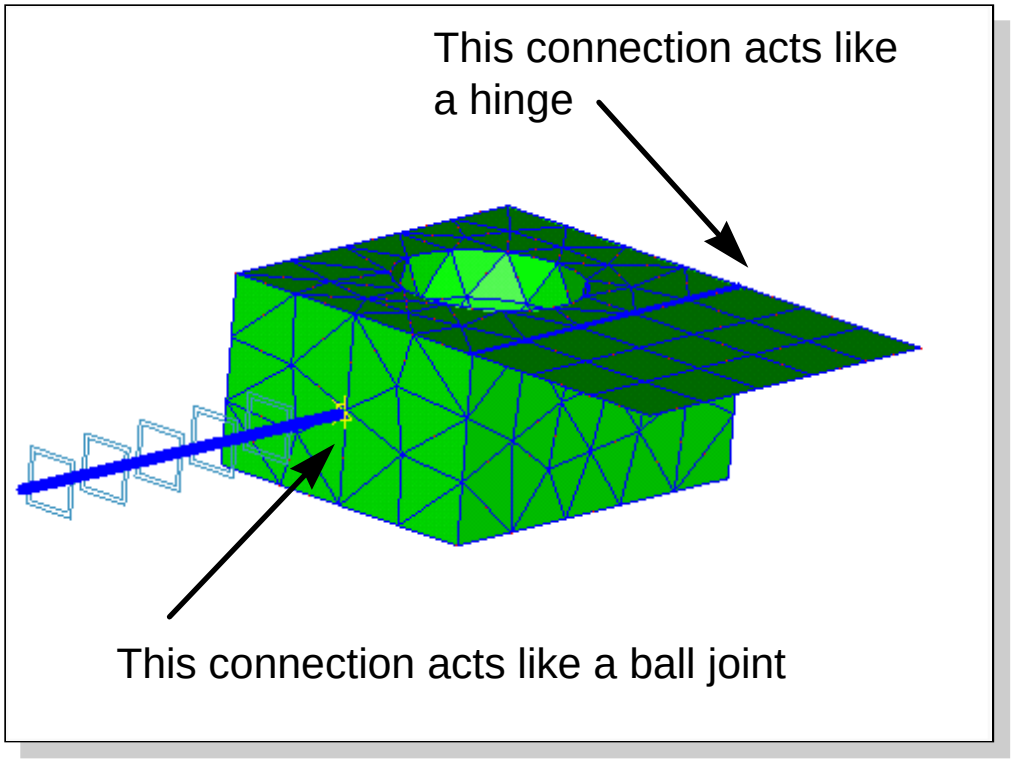
Modify spring elem, select Family and change it to Gap



Connecting Beams, Shells to Solids

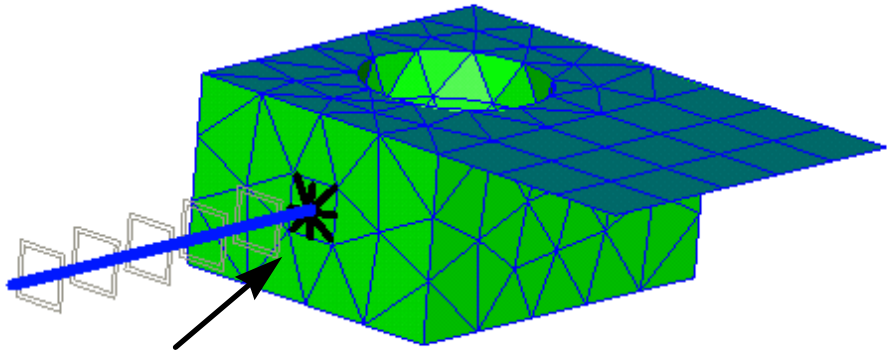
- Solid elements have 3 dof, shells and beams have 6 dof
- Beams connected to solids will act like a ball joint connection
- Shells connected to solids along colinear nodes will act like hinge
- Don’t let Model Solution put in springs, you’re loosing control
- Sometimes it is tempting to use rigid bars to “glue” solid elements together, don’t do this
- Techniques
  - Embed elements into solids
  - Surface coat solids with 6 dof shells
  - Use rigid elements to “spider” to adjacent nodes
  - Use constraint elements as a soft “spider”

Connecting Beams, Shells to Solids



Connecting Beams, Shells to Solids

Surface coat adjacent surface
with shells of similar thickness

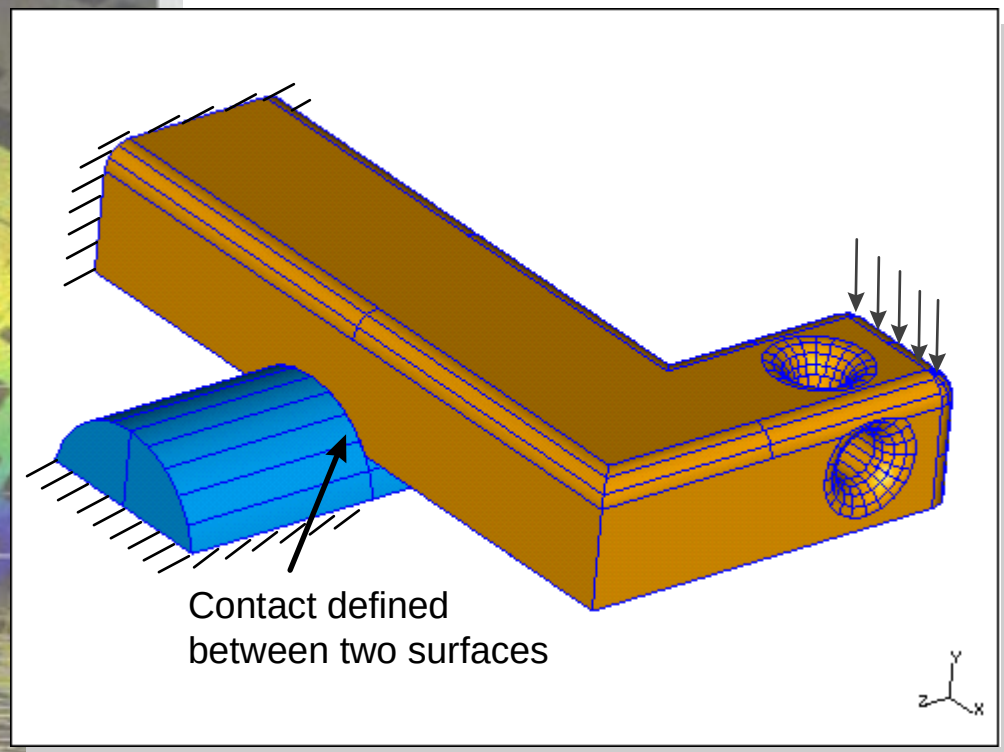


"Spider" rigid or constraint element to
footprint scribed on solid model
"Split Surface" is useful for this

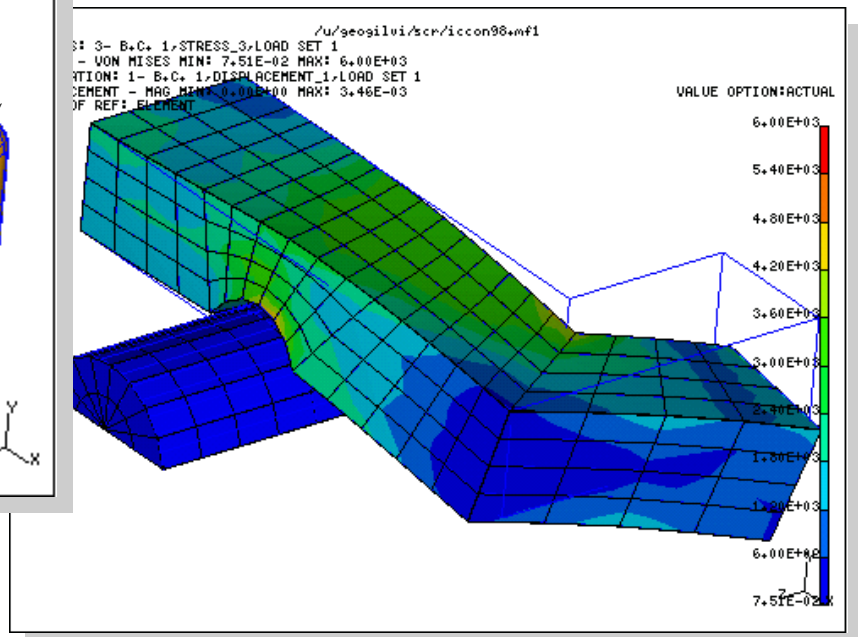
Breakout Modeling

- Substructure analysis using Data Surfaces
- Model large structures coarsely, solve, then apply displacements to breakout
- Technique
  - Need to be geometry based at interface to use data surfaces
  - Create partitions at interfaces, solve coarse model
  - Create breakout as second FE model, create data surfaces from results
  - Need to apply X Y and Z displacements individually
  - Data edges can be used for shells
  - Shouldn't use when critical stresses are near the breakout location

Breakout Modeling

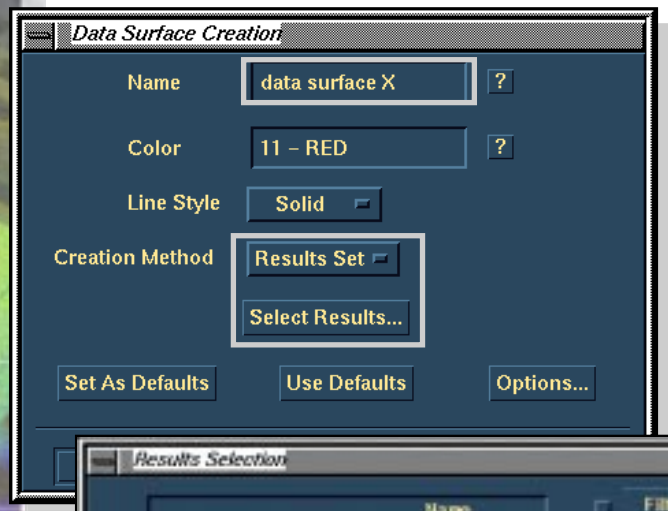


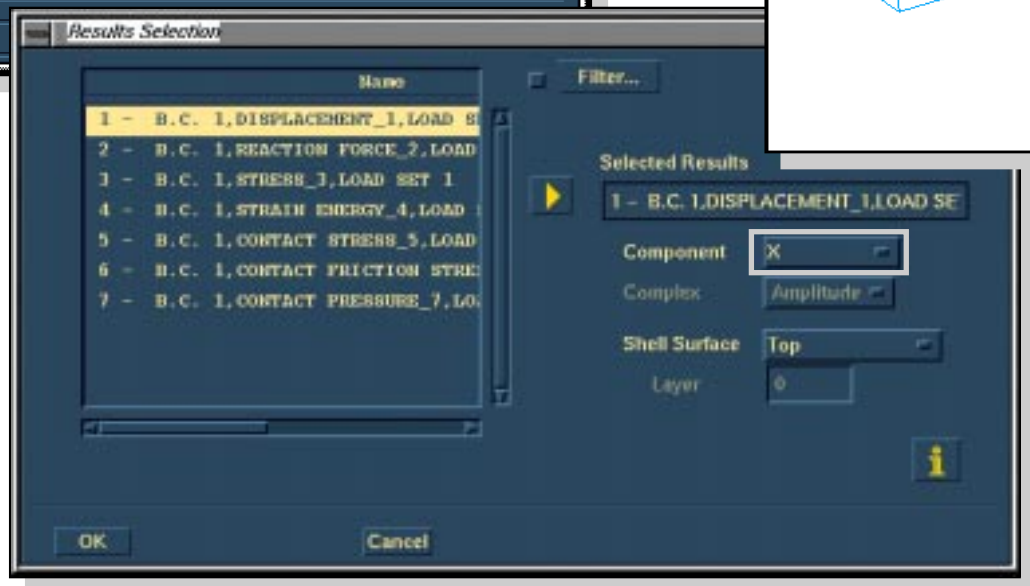
Problem: You want to do contact analysis using bricks and then do detailed analysis with iterative solver and parabolic tets

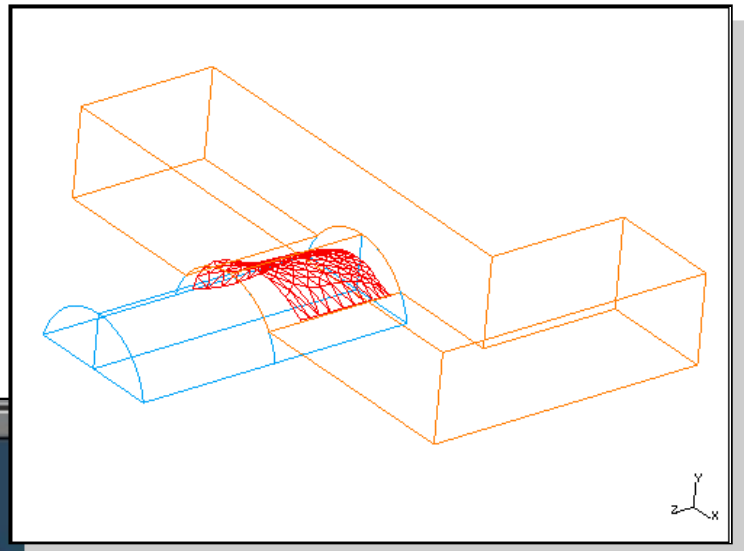


First step, suppress details, generate mesh and solve

Breakout Modeling



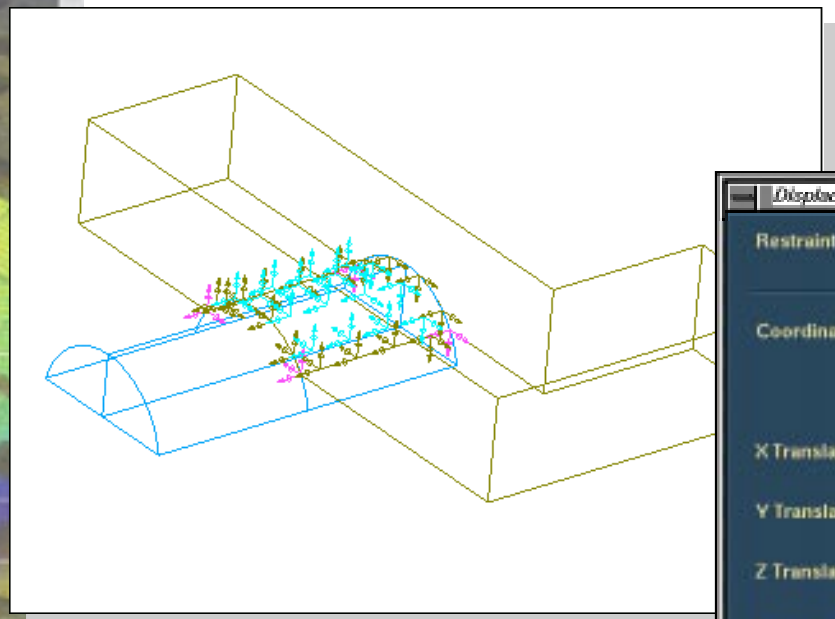


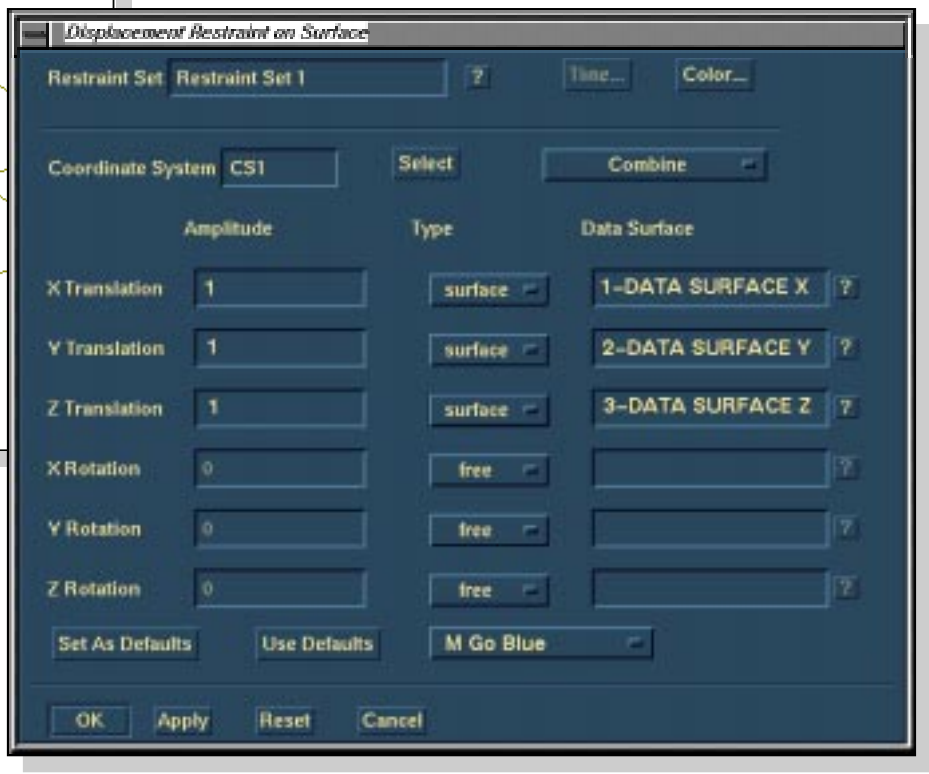


Next step, create new FE model on same part and create datasurfs from results to map displacements onto new model

You need to create a datasurf for each displacement component

Breakout Modeling

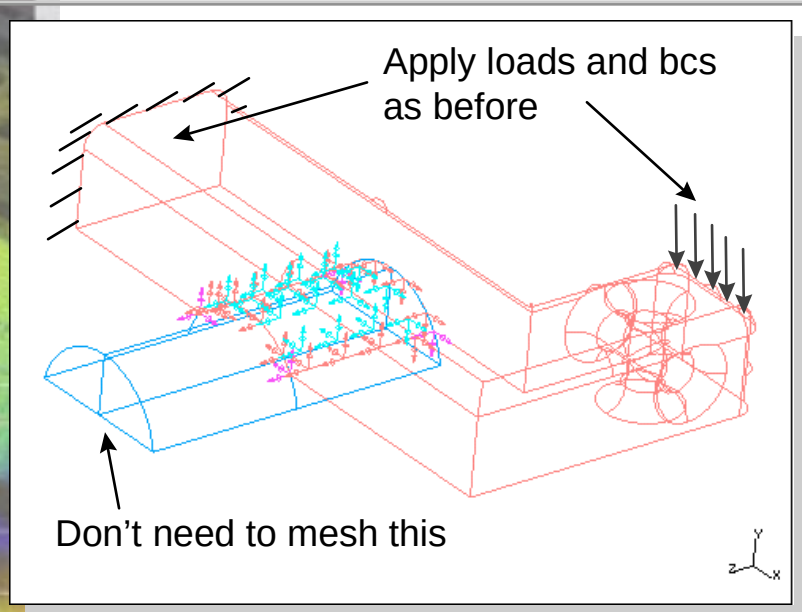


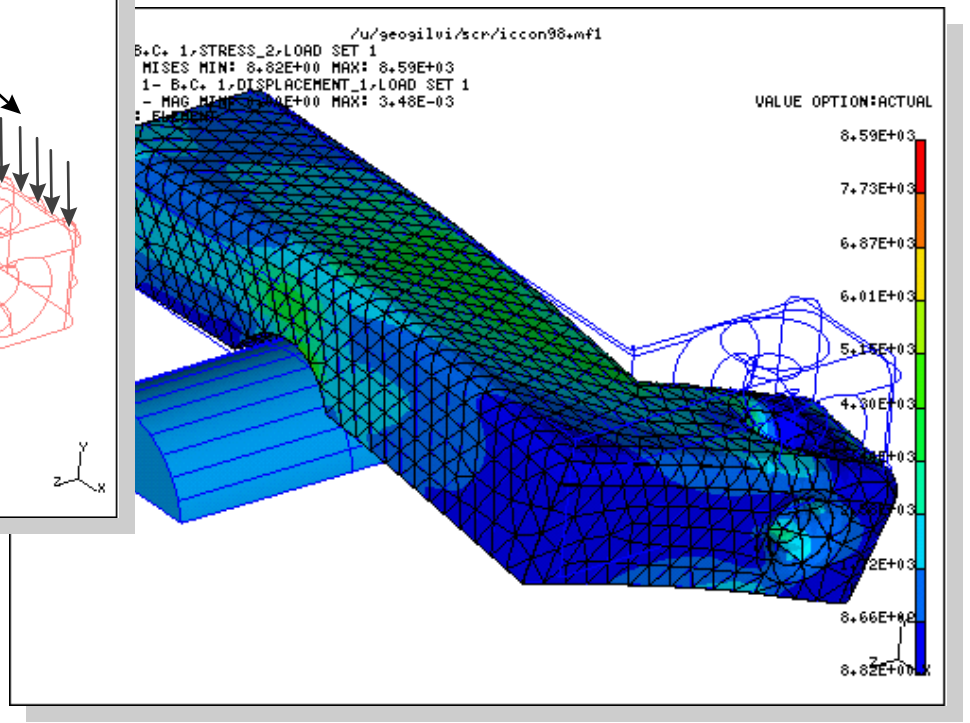


Use the data surfaces to apply enforced displacements on the surface

Note that you have to use different data surfaces for the X Y and Z displacements

Breakout Modeling





Next step, unsuppress features and update part, restraints should survive update if surface doesn’t change

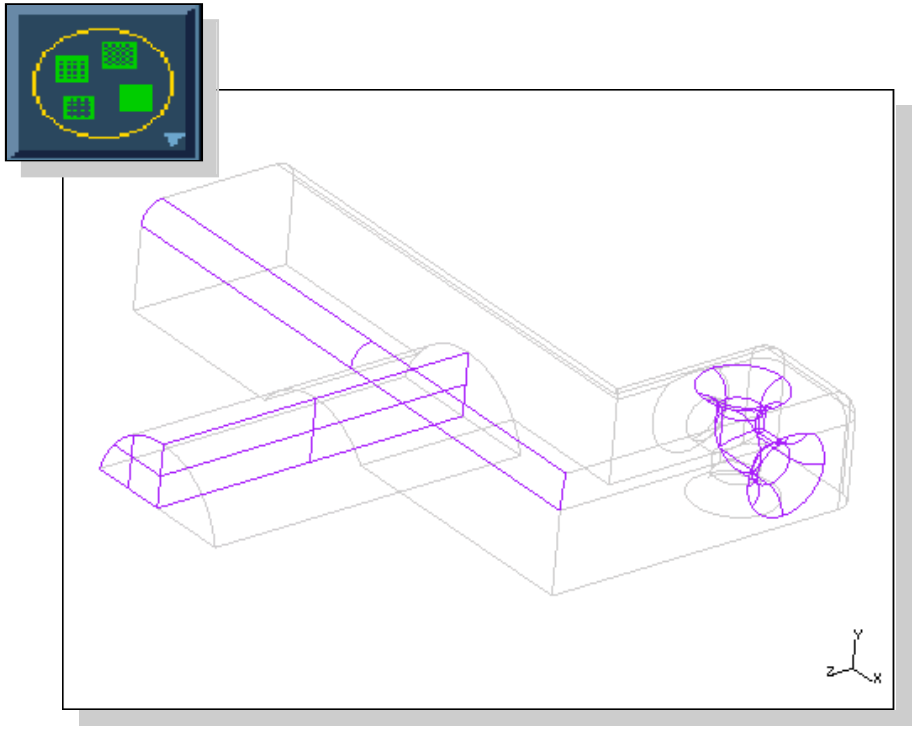
Important point, to do this, you have to copy part and FE model into a new part **>without the results<**

If you aren’t changing geometry, you can remesh on same part

>Important< you can only use the original load vector with this technique

Breakout Modeling

- Tip - To suppress many features at a time, put feature surfaces into a design group and use RMB “Use Design Groups…”



This has to be done in Design Task :~(

Create Design Group with one surface for each feature

Modify part, instead of picking part, use RMB and filter for Feature only **>Important<**

Now, use RMB again and pick “Use Design Groups…”

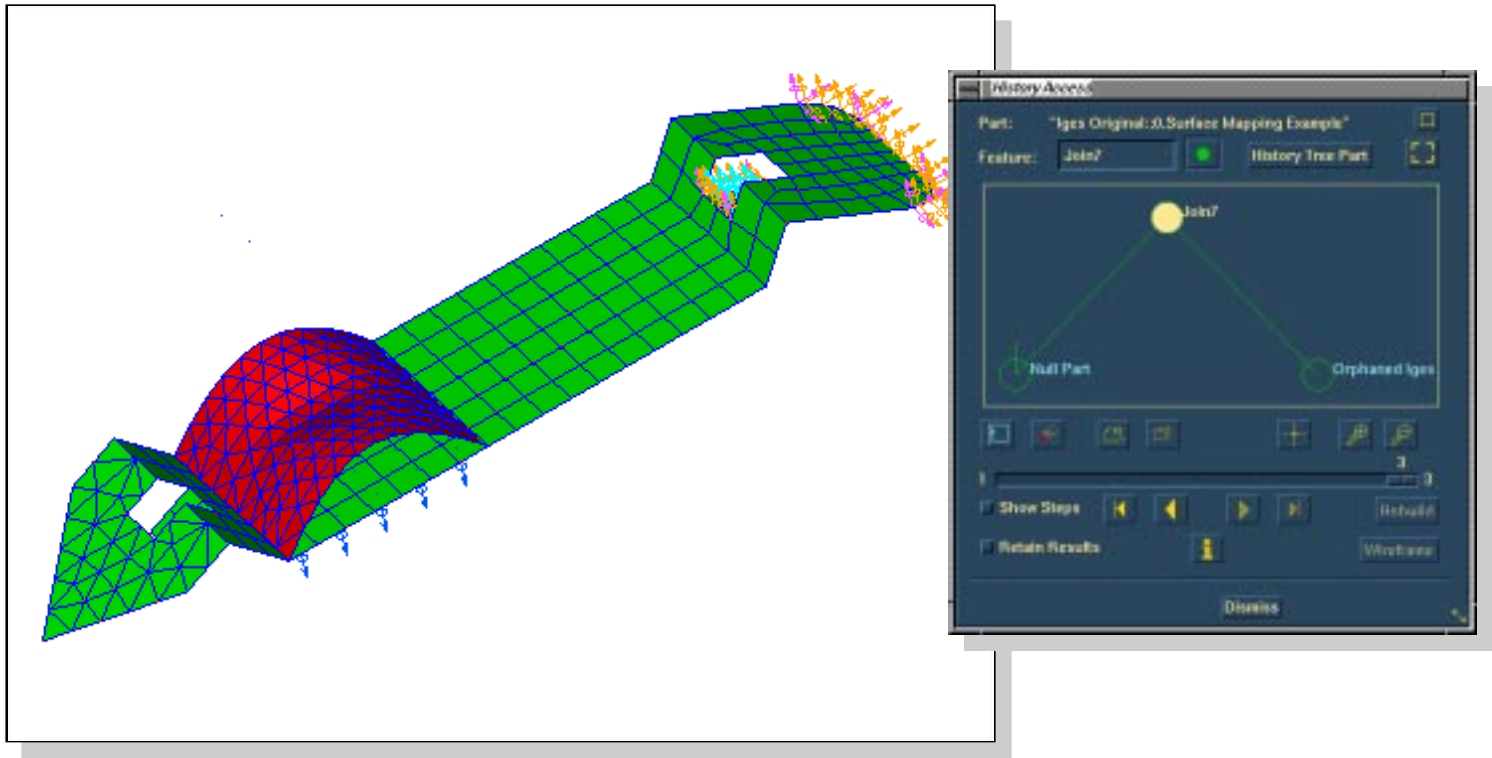
Picking the Design Group will select all the associated features and allow you to suppress them in one go

Feature Replacement

- What happens if you build on imported geometry and the geometry changes?
- Technique
  - Feature replacement allows you to swap out geometry and still keep geom based FE stuff
  - Maps surfaces of old geometry on to new

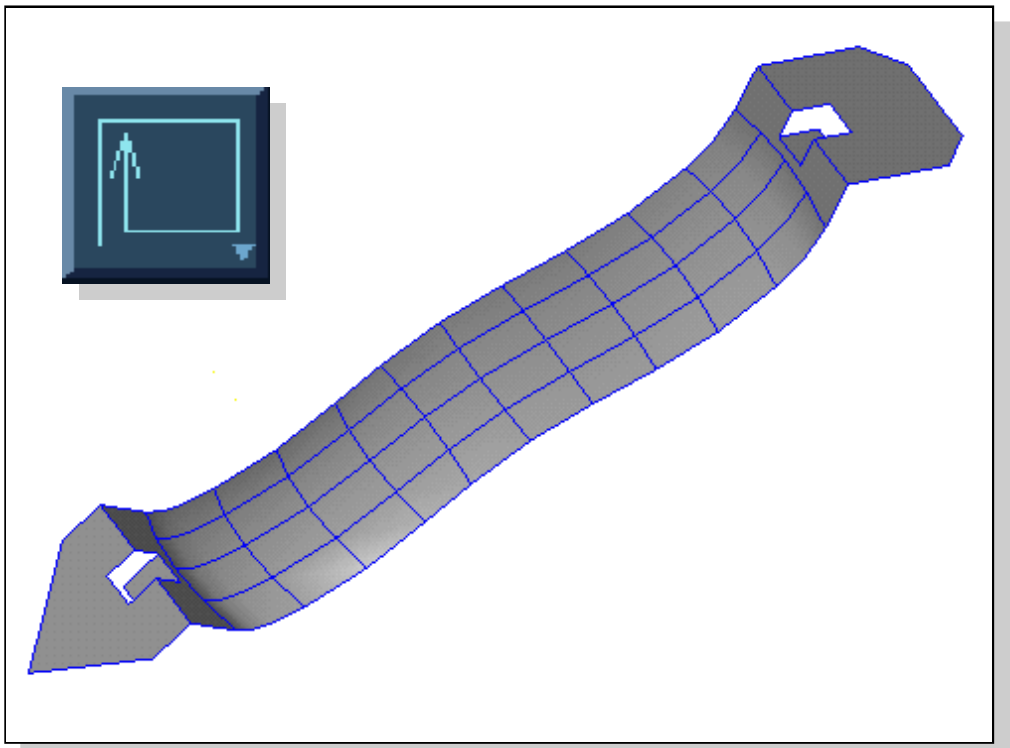
Feature Replacement

- Example, import Iges geometry, join this to null part, add FE infomation

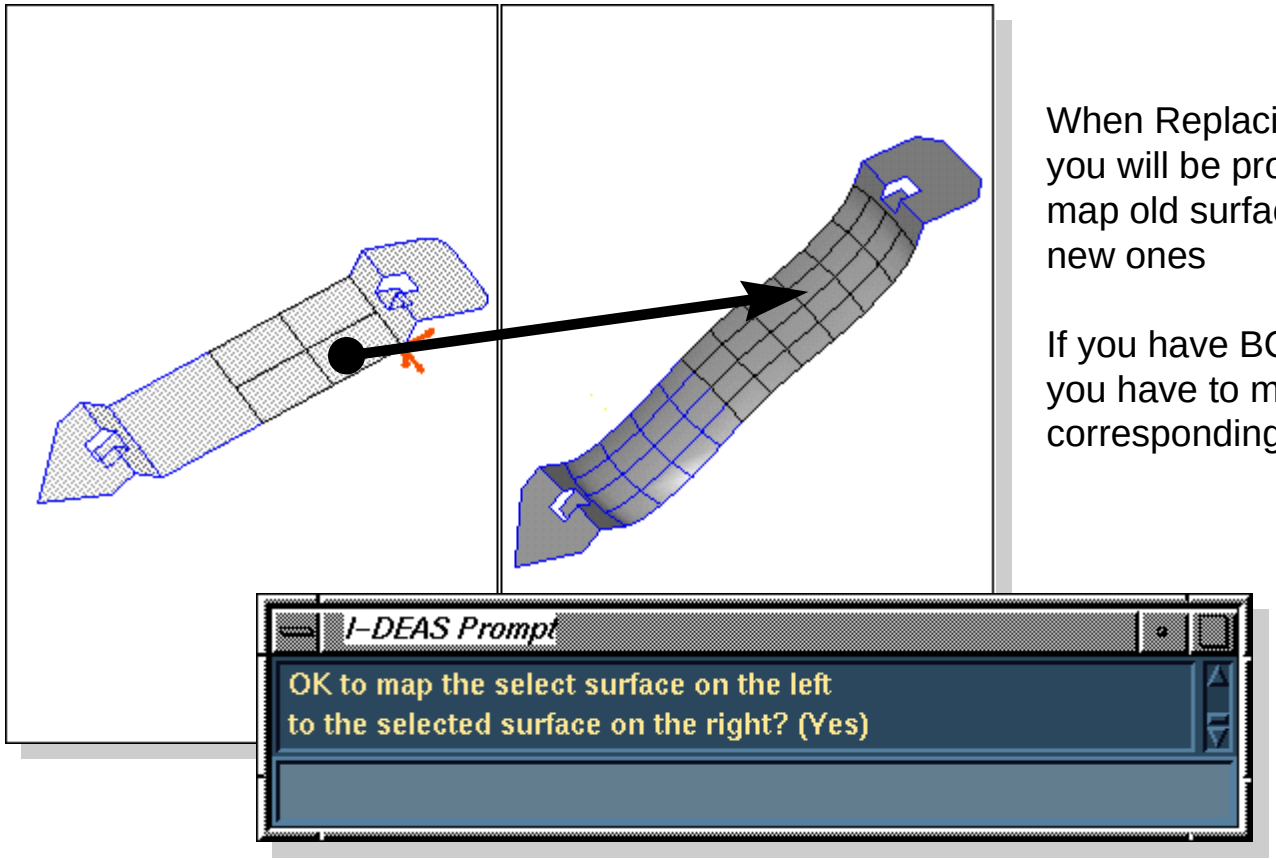


Feature Replacement

- Industrial Designer thinks it will be cool to restyle part, you get a new Iges file
- You can use “Replace Feature” in Master Modeler to remap FE information onto new geometry, invoke by picking on feature and modifying



Feature Replacement

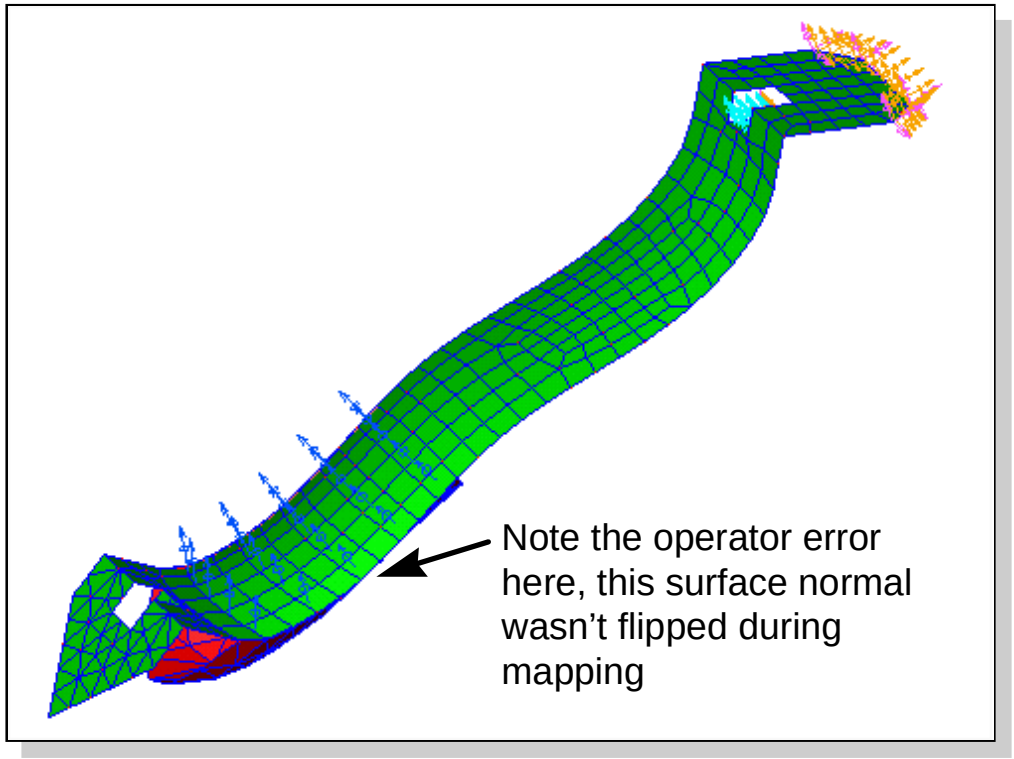


When Replacing Features you will be prompted to map old surfaces onto new ones

If you have BCs on edges you have to map the corresponding edges

Feature Replacement

- Update Part, Mesh Defs and BCs transfer to new feature

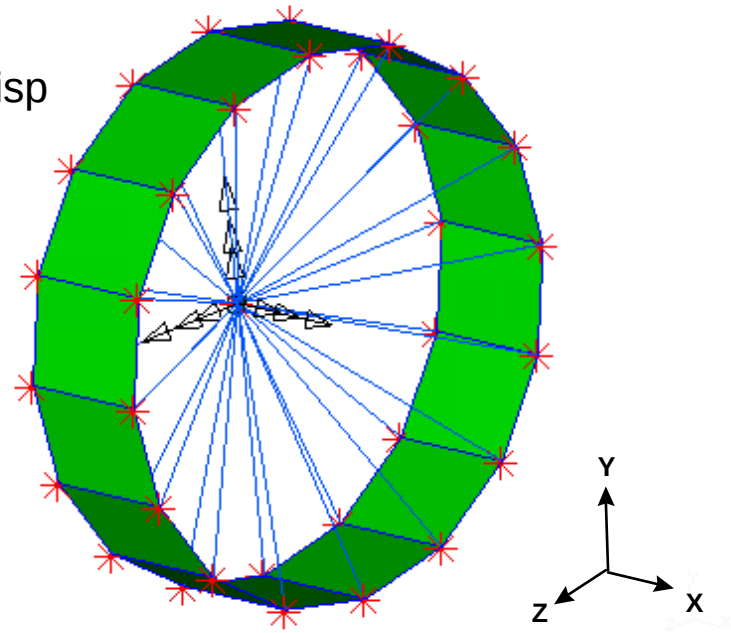


The Amazing Xpanding Rigid Element

- Create squirrel cage, use rigid element as spokes
- Give wheel rotational displacement, no loads
- Post process displacements, looks like it's expanding, what gives?
- Happens because of small displacement theory behind linear analysis
- Displacement are tangent, don't follow path
- Occurs in dynamic analyses where there may be localized rotational rigid body motion

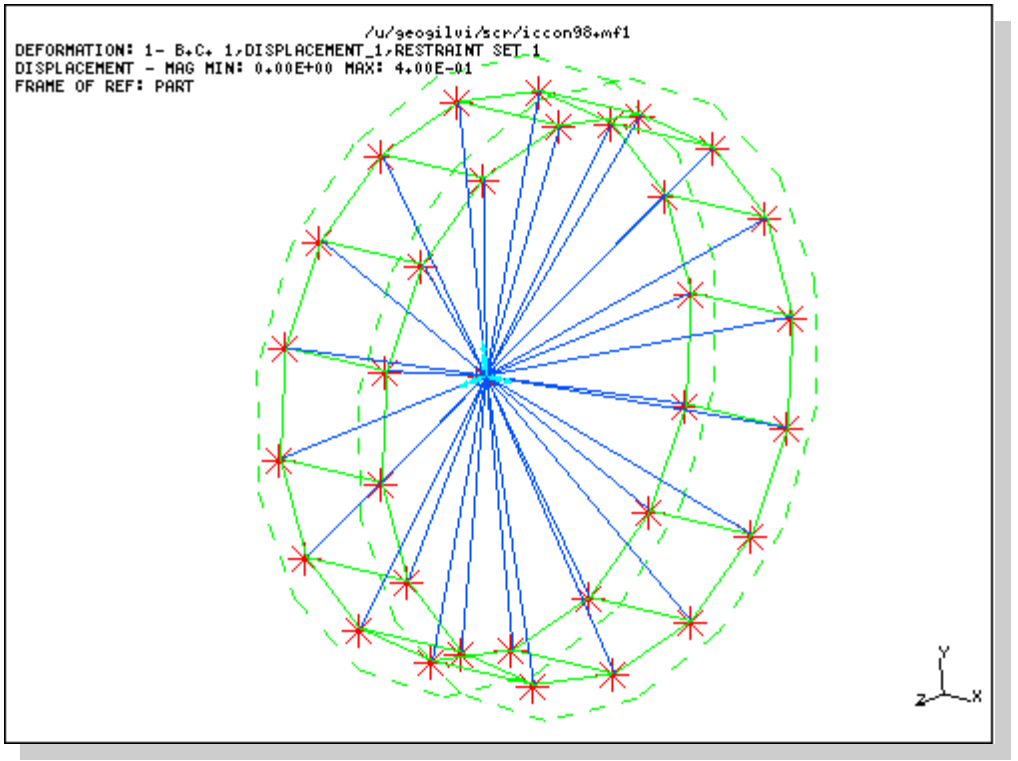
The Amazing Xpanding Rigid Element

Enforced Disp
at hub
X=0
Y=0
Z=0
RX=.1
RY=0
RZ=0



Model of squirrel cage, spokes are rigid elements
Radius = 4

The Amazing Xpanding Rigid Element

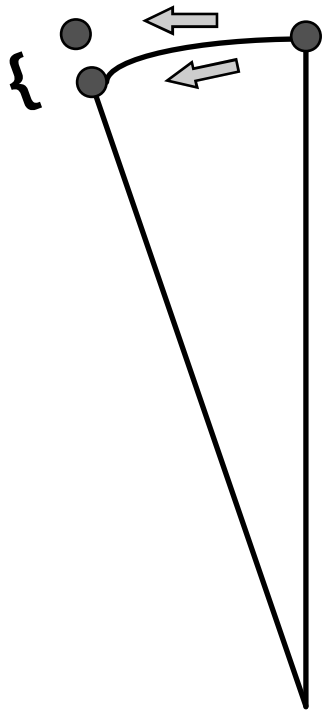


Post Process displacements, it looks like it's expanding

The Amazing Xpanding Rigid Element

Reason: Linear Analysis is based on small displacement theory where Sin (θ) = θ, displacements follow tangent direction instead of the path

Difference is the apparent expansion



Note: you would see the same behaviour in Model Solution Geometric Non-Linear, the rigid elements don't update

Other Resources

- SDRC WWW Simulation Tech Tips
  - www.sdrc.com
  - Look under “Tech Info” on the home page
- NAFEMS
  - www.nafems.org
  - FE certification and training

Get
There
Faster™