Topology and Shape Optimization

Università di Firenze
Dipartimento di Meccanica e Tecnologie Industriali

5 Novembre 2012

Ing. Mauro Linari
Project Manager
MSC Software S.r.l.

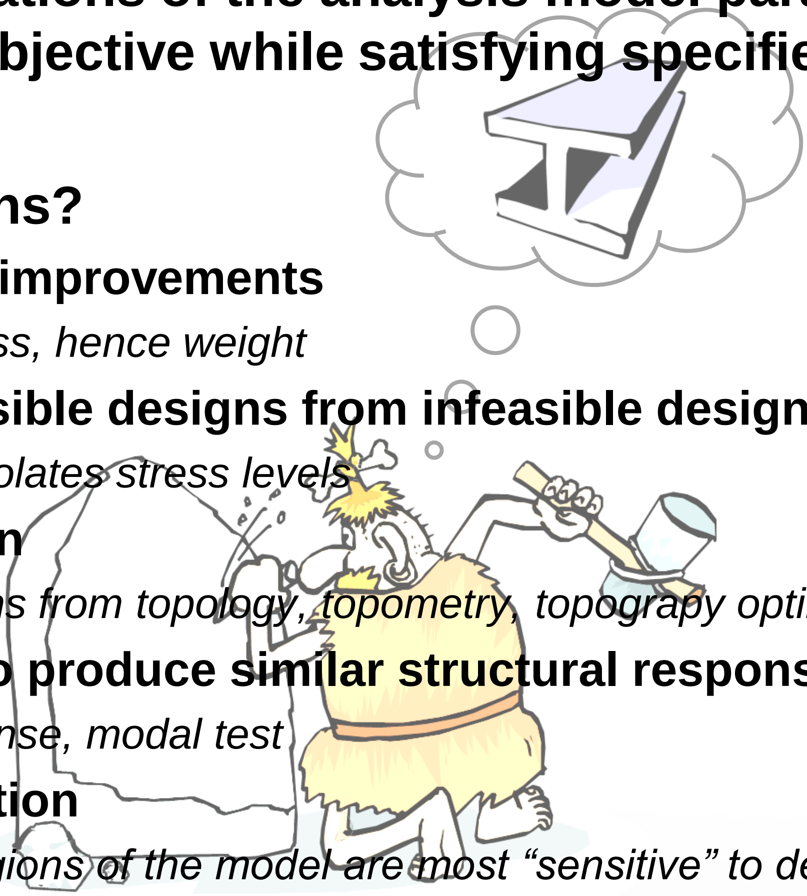
MSC Software
Simulating Reality, Delivering Certainty™

Topology and Shape Optimization
Agenda

- **Brief introduction to Structural Optimization in MSC Nastran**
  - Sizing Optimization
  - Topometry Optimization
- **Topology Optimization**
- **Shape Optimization**
  - Brief Introduction to Topography Optimization

Topology and Shape Optimization
What is “DESIGN OPTIMIZATION?

- **Automated modifications of the analysis model parameters to achieve a desired objective while satisfying specified design requirements.**
- **Possible applications?**
  - **Structural design improvements**
    - *Minimize thickness, hence weight*
  - **Generation of feasible designs from infeasible designs**
    - *Original model violates stress levels*
  - **Preliminary Design**
    - *Candidate designs from topology, topometry, topograpy optimization*
  - **Model matching to produce similar structural responses**
    - *Frequency response, modal test*
  - **Sensitivity evaluation**
    - *Identify which regions of the model are most “sensitive” to design changes or imperfections*
    - *System parameter identification*



Classes of "Design Optimization"

Which classes are implemented in MSC Nastran?

SIZING OPTIMIZATION

Element, Material and Connectivity properties can be used as design variables

TOPOMETRY OPTIMIZATION

Special automated procedure for sizing optimization

SHAPE OPTIMIZATION

The shape of the structure can be varies to satisfy specific requirements

TOPOGRAPHY OPTIMIZATION

Special shape optimization procedure in which bead Theory is used to modify the shape of the structure

TOPOLOGY OPTIMIZATION

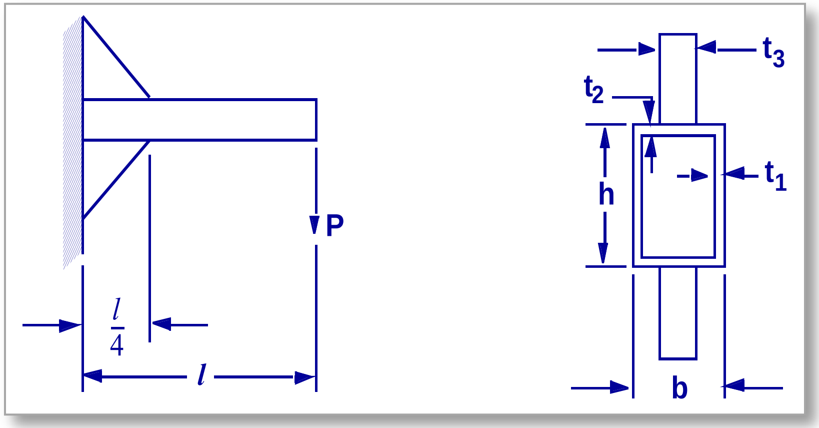
It finds an optimal distribution of material given the package space, loads and boundary conditions

Design Optimization Overview

Optimization problem statement

- **Design Variables:**
  - Find {X} = { X1, X2, …, XN }
    - e.g., thickness of a panel, area of a stiffener
- **Objective Function:**
  - Minimize F(X)
    - e.g., weight
- **Subject to:**
  - Inequality constraints:
    - $G_j(X) \leq 0$ j = 1,2,….,L
    - Design Criteria and margins
  - Side constraints:
    - $X_i^L \leq X_i \leq X_i^U$ i = 1,2,….,N
    - Gage allowables

Gusseted Tube Connection



- ***Minimize structural mass with constraints on:***
  - ***Stresses in the gussets:***

    $$\sigma_{\min plate} \leq \sigma_{plate} \leq \sigma_{\max plate}$$

  - ***Buckling stresses in the tube:***

    $$\sigma_{tube} \leq \sigma_{buck}$$

  - ***Sizing and gauge requirements:***

    $$t_1 \geq t_{1\min},\ t_2 \geq t_{2\min},\ t_3 \geq t_{3\min}$$

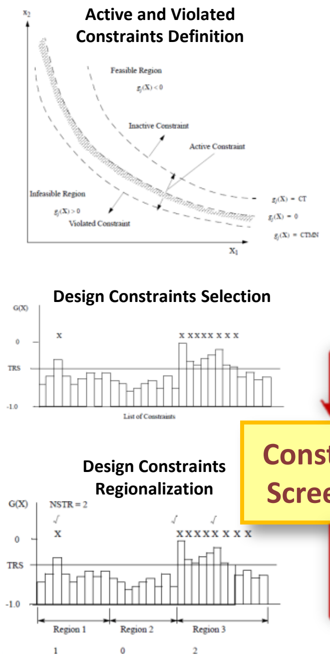
    $$2t_1 \leq b,\ 2t_2 \leq h,\ t_3 \leq b$$

Design Model ⇒ *Design Variables + Constraint Functions + Objective Functions*

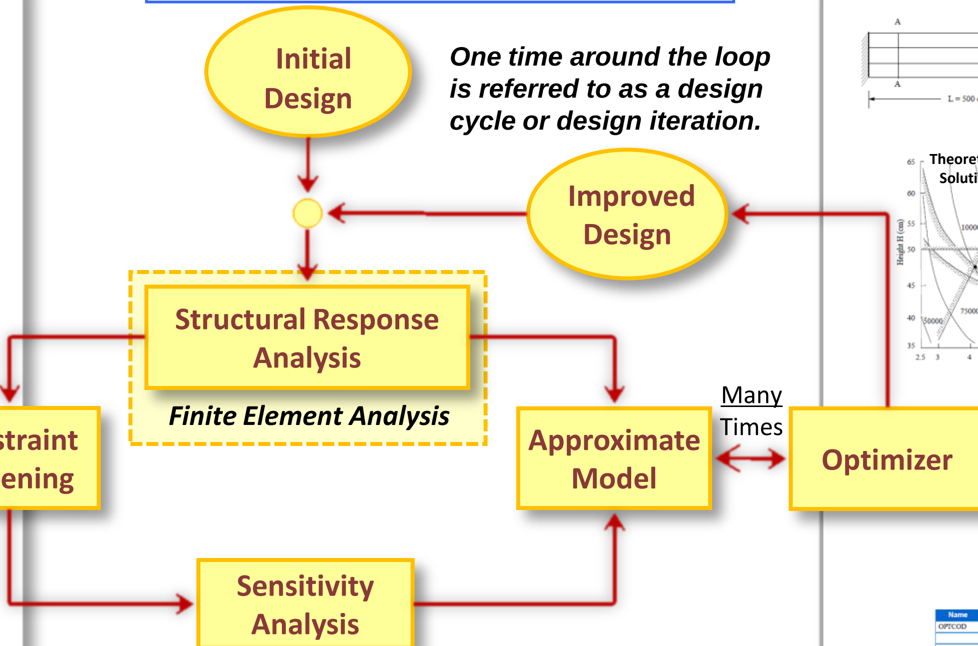
Analysis Model ⇒ *Finite element model*

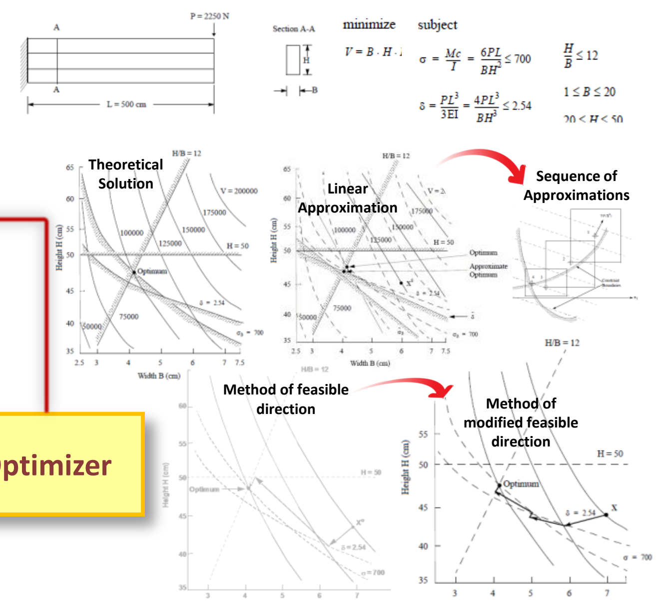
Design Optimization Overview

MSC Nastran Implementation of Structural Optimization



Based on sensitivities information



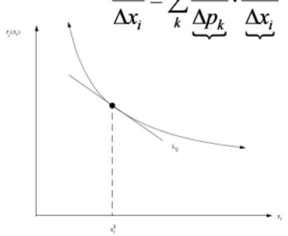


Name	Description, Type and Default Value	
OPTCOD	OPTCOD (Character; Default = Blank)	
	= Blank	taken from system cell number 413
	= "MSCADS"	MSCADS is used
	= "IPOPT"	IPOPT is used

Name	Description, Type and Default Value	
METHOD	Optimization method (Integer ≥ 0; Default = 0)	
	0	Automatic selection for a better performance based on number of design variables, number of constraints, number of active/violated constraints and computer memory.
	1	Modified Method of Feasible Directions for MSCADS.
	2	Sequential Linear Programming for MSCADS
	3	Sequential Quadratic Programming for MSCADS
	4	SUMT method for MSCADS
	IJK	See Table 4-9

Design Sensitivity terms are calculated only for the selected design constraints

$$\frac{\Delta r_j}{\Delta x_i} = \sum_k \underbrace{\frac{\Delta r_j}{\Delta p_k}} \cdot \underbrace{\frac{\Delta p_k}{\Delta x_i}}$$

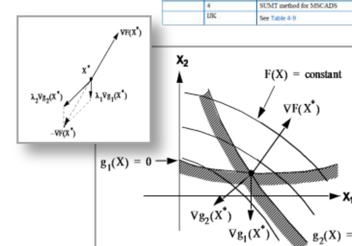


- **A 'Brute Force' coupling of an analysis program and an optimization program suffers from:**
  - Too many design variables
  - Too many design constraints
  - Too many detailed analysis

Finite Element Analysis Program ⟷ Numerical Optimization Program

- **The Approximate Model now acts as the interface between the analysis and the optimizer**
  - It contains inexpensive and easily computed yet high-quality, explicit approximations of the objective function and all retained constraints.
  - It includes the results of constraint deletion.
  - It is updated by a full finite element analysis every design cycle

$$f(X^0+\Delta x) = f(X^0) + \left.\frac{df}{dx}\right|_{X^0}\cdot\Delta x + \left.\frac{d^2f}{dx^2}\right|_{X^0}\cdot\frac{\Delta x^2}{2!} + \left.\frac{d^3f}{dx^3}\right|_{X^0}\cdot\frac{\Delta x^3}{3!} + \dots \Longrightarrow \tilde{f}(X^0+\Delta x) = f(X^0) + \left.\frac{df}{dx}\right|_{X^0}\cdot\Delta x$$



KUHN-TUCKER Condition for Constrained Optimization Optimum Condition (Graphical Representation)

Design Optimization Overview

Design Model Definition

- **Flowchart for Design Modeling**

Define Analysis Disciplines
Executive Control: **SOL 200**
Case Control: **Analysis = ...**

Define Design Variables
Bulk Data: **DESVAR**

Relate Design Variables to Analysis Model:
Bulk Data: **DVPREL1, DVCREL1, DVMREL1**
DVPREL2, DVCREL2, DVMREL2

$$\left[\begin{array}{c} Analysis \\ Model \\ Parameters \end{array}\right] = \left[\begin{array}{c} P_L = [P_0] + [C_2].[X] \\ \hline P_{NL} = [f(X,C)] \end{array}\right] \begin{array}{l} \leftarrow \text{DV}x\text{REL1} \\ \leftarrow \text{DV}x\text{REL2} \end{array}$$

Define Design Responses
Bulk Data: **DRESP1**
DRESP2

Define Objective
Case Control: **DESOBJ**

Define Constraints
Case Control: **DESGLB, DESSUB**
Bulk Data: **DCONSTR, DCONADD**

Provide Optional Parameters
Bulk Data: **DOPTPRM**

Design Optimization Overview

Design Response Type 1 (DRESP1)

- **Prior to defining the objective and constraint functions, the responses on which they are based need to be identified.**
  - Type 1 – Responses obtained directly from an MD Nastran analysis

Responses Type (RTYPE)	Responses Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTi (Integer > 0)
Weight	Row Number (1 … See Remark 25		
VOLUME	Blank		
EIGN	Real Eigenvalue … Mode Number (I…		
CEIG	Complex Eigenv… Number (Integer…		
FREQ	Real Eigenvalue … Number (See Re…		
LAMA	Buckling Mode N…		
DISP	Displacement C…		
STRAIN	Strain Item Code…		
ESE	Strain Energy Ite… See Remark 20.		
STRESS	Stress Item Code…		

Responses Type (RTYPE)	Responses Attributes		
	ATTA (Integer > 0)	ATTB (Int. > 0 or Real > 0.0)	ATTi (Integer > 0)
FORCE	Force Item Code	Blank	Property Entry (PID)
SPCFORCE	SPC Force Item …		
CSTRAIN	Strain Item Code…		
CSTRESS	Stress Item Code…		
CFAILURE	Failure Criterion…		
CSTRAT	Composite Stren… Ratio Item Code…		
TOTSE (Total Strain Energy)	Blank		
GPFORCE	Grid Point (See … 25).		

Responses Type (RTYPE)	Responses Attributes		
	ATTA (Integer > 0)	ATTB (Int. > 0 or Real > 0.0)	ATTi (Integer > 0)
GPFORCP	Grid Point (See Remark 25.)	Blank	GRID ID connected to ATTA
FRACCL	Acceleration C…		
FRDISP	Displacement …		
PRES	Acoustic Press… Component		
FRFORCE	Force Item Co…		
FRSPCF	SPC Force Co…		

Responses Type (RTYPE)	Responses Attributes		
	ATTA (Integer > 0)	ATTB (Int. > 0 or Real > 0.0)	ATTi (Integer > 0)
PSDDISP	Displacement Component … Remark 27)	Frequency Value (Blank, …	Grid ID
PSDVELO	Velocity Component … Remark 27)		
PSDACCL	Acceleration Component … Remark 27)		
RMSDISP	Displacemen… Component		
RMSVELO	Velocity Com…		
RMSACCL	Acceleration Component		
TDISP	Displacemen… Component		

Responses Type (RTYPE)	Responses Attributes		
	ATTA (Integer > 0)	ATTB (Int. > 0 or Real > 0.0)	ATTi (Integer > 0)
TVELO	Velocity Component	Time Value (Real) See Remark 16.	GRID ID
TACCL	Acceleration Component	Time Value (Real) See Remark 16.	GRID ID
TSPCF	SPC Force Component	Time Value (Real) See Remark 16.	GRID ID
TSTRE	Stress Item Code	Time Value (Real) See Remark 16.	Property Entry (PID)
TFORC	Force Item Code	Time Value (Real) See Remark 16.	Property Entry (PID)
TRIM	AESTAT or AESURF Entry ID	Blank	Blank
STABDER			
FLUTTER			

Responses Type (RTYPE)	Responses Attributes		
	ATTA (Integer > 0)	ATTB (Int. > 0 or Real > 0.0)	ATTi (Integer > 0)
FRSTRE	Stress Item Code	Frequency Value (Real > 0.0). See Remark 15.	Property Entry (PID)
FRVELO	Velocity Component	Frequency Value (Real > 0.0). See Remark 15.	GRID ID

$$[W]=\begin{bmatrix} W_x & W_{12} & W_{13} & W_{14} & W_{15} & W_{16} \\ W_{21} & W_y & W_{23} & W_{24} & W_{25} & W_{26} \\ W_{31} & W_{32} & W_z & W_{34} & W_{35} & W_{36} \\ W_{41} & W_{42} & W_{43} & I_x & W_{45} & W_{46} \\ W_{51} & W_{52} & W_{53} & W_{54} & I_y & W_{56} \\ W_{61} & W_{62} & W_{63} & W_{64} & W_{65} & I_z \end{bmatrix}_{6\times6}$$

MAX, MIN

$$SUM(X_1,X_2,...,Xn)=\sum_{i=1}^{n}X_i$$

$$AVG(X_1,X_2,...,Xn)=\sum_{i=1}^{n}X_i/n$$

$$SSQ(X_1,X_2,...,Xn)=\sum_{i=1}^{n}X_i^2$$

$$RSS(X_1,X_2,...,Xn)=\sqrt{\sum_{i=1}^{n}X_i^2}$$

Design Optimization Overview
Design Response Type 1 (DRESP1)

DRESP1 **Design Sensitivity Response Quantities**

Defines a set of structural responses that is used in the design either as constraints or as an objective.

Format:

1	2	3	4	5	6	7	8	9	10
DRESP1	ID	LABEL	RTYPE	PTYPE	REGION	ATTA	ATTB	ATT1	
	ATT2	-etc.-							

Example:

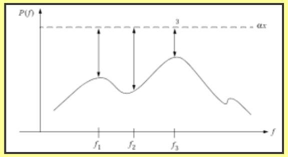
DRESP1	1	DX1	STRESS	PROD	2	3		102	
	103								

Field	Contents
ID	Unique entry identifier. (Integer > 0)
LABEL	User-defined label. (Character)
RTYPE	Response type. See Table 8-9. (Character)
PTYPE	Element flag (PTYPE = "ELEM") or property entry name. Used with element type responses (stress, strain, force, etc.) to identify the property type, since property entry IDs are not unique across property types. (Character: "ELEM", "PBAR", "PSHELL", etc.)
REGION	Region identifier for constraint screening. See Remark 10. for defaults. (Integer > 0)
ATTA, ATTB, ATTi	Response attributes. See Table 8-9. (Integer > 0 or Real or blank)

Design Optimization Overview
Design Response Type 2 (DRESP2)

- **Type 2 (Second-level response)**
- **Responses written using equations**
  - They can be written as function of:
    - Design Variables
    - Table Constants
    - Type 1 Responses
- **Examples**
  - Generation of a new stress or strain failure criterion that is not available in MD.Nastran
  - Imposing local buckling criteria based on element sizes as well as stress components
  - Programming proprietary design-sizing equations
  - Avoid a frequency range.

FUNC=BETA facilitates a design task where the objective is to minimize the maximum response. It does this by creating the following design task:



Minimize $\phi = C_1 X_\beta$

Subject to $g_j = \dfrac{r_j - \gamma X_\beta}{C_3}$

where γ is determined from $C_2 = \dfrac{r_{j\max} - \gamma X_\beta}{C_3}$

FUNC=MATCH facilitates a design task where the objective is to minimize the difference between analysis results :

When ***METHOD = LS***, a least square minimization is performed where the objective is:

$$\phi_j = \sum_{j=1}^{m} \left(\frac{r_j - r^T{}_j}{r^T{}_j} \right)^2$$

When ***METHOD=BETA***, the design task becomes one of minimizing the maximum normalized difference between the analysis and target values

$$\frac{r_j - r^T{}_j}{\left| r^T{}_j \right|}$$

EXAMPLE - RESPONSE FOR CENTERS OF GRAVITY (CG)

1. Create two responses for the locations of Center of Gravity, X_{cg} and Y_{cg}
2. Assume the same weight in x and y directions. The CG locations are defined by

$$X_{cg} = \frac{W_{26}}{W_{33}}, \quad Y_{cg} = \frac{W_{34}}{W_{33}}$$

(W_{26}, W_{34} and W_{33} must be previously defined as *Type 1* response)

Design Optimization Overview
Design Response Type 2 (DRESP2)

Format:

1	2	3	4	5	6	7	8	9	10
DRESP2	ID	LABEL	EQID or FUNC	REGION	METHOD	C1	C2	C3	
	"DESVAR"	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	"DTABLE"	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						
	"DRESP1"	NR1	NR2	NR3	NR4	NR5	NR6	NR7	
		NR8	-etc.-						
	"DNODE"	G1	CMM	G2	CMP2	G3	CMP3		
		G4	C4	etc.					
	"DVPREL1"	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7	
		DPIP8	DPIP9	-etc.-					
	"DVCREL1"	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7	
		DCIC8	DCIC9	-etc.-					
	"DVMREL1"	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7	
		DMIM8	DMIM9	-etc.-					
	"DVPREL2"	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7	
		DPI2P8	DPI2P9	-etc.-					
	"DVCREL2"	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7	
		DCI2C8	DCI2C9	-etc.-					
	"DVMREL2"	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7	
		DMI2M8	DMI2M9	-etc.-					
	"DRESP2"	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7	
		NRR8	-etc.-						

DEQATN **Equation Definition**

Defines one or more equations for use in analysis.

Format:

1	2	3	4	5	6	7	8	9	10
DEQATN	EQID	EQUATION							
	EQUATION (Cont.)								

Example:

DEQATN	14	F1(A, B, C, D, R) = A + B · C – (D ** 3 + 10.0) + sin(PI(1) · R)							
	+ A**2 / (B – C) ; F = A + B – F1 · D								

Field	Contents
EQID	Unique equation identification number. (Integer > 0)
EQUATION	Equation(s). See Remarks. (Character)

Design Optimization Overview

Design Response Type 2 (DRESP2) – Avoiding a frequency range

- **Often it is important to design a structure to avoid certain frequency ranges to reduce dynamic amplification at resonance:**
  - Known forcing function
  - Known interaction (bending/torsion or coupling with other structure)
- **The following algebra equation can be used to keep the frequency away from a given frequency range:**

If **f < fl** or **f > fu**, and **fl<fu** then;

(f-(fl+ fu)/2)2 > ((fu- fl)/2)**2**

Where:
f = calculated frequency
fl, = lower bound frequency
fu = upper bound frequency

- **The first natural frequency should be outside the range of 30Hz and 34Hz:**

Initial Freq.	**Desired Freq.**	
F1 = 31.8Hz	F1<30	F1>34

- **Fstar=(f-(fl+ fu)/2)**2 is defined using DRESP2/DEQATN.**
- **The lower limit: Fstar>4=((34-30)/2))**2 is imposed with DCONSTR**

```
DRESP1    102          F1        FREQ                      1
DRESP2  151          Fstar     99
           DTABLE  f1l          f1u
           DRESP1 102
DEQATN    99          Fstar(fl,fu,f) = (f-(fl+fu)/2.)**2
DTABLE    f1l          30.         F1u      34.
DCONSTR 201          151          4.
```

Design Optimization Overview

Design Response Type 2 (DRESP2) – Response for center of gravity

- **Create two responses for the locations of Center of Gravity, X_{cg} and Y_{cg}.**
- **Assume the same weight in x and y directions. The CG locations are defined by**

$$X_{cg} = \frac{W_{26}}{W_{33}}, \; Y_{cg} = \frac{W_{34}}{W_{33}}$$

- **They are defined in two steps:**
  - Create three DRESP1 entries 41, 42, 43 referring to W_{33}, W_{26} and W_{34}
  - Create two DRESP2 entries 50, 51 and DEQATN 200 for X_{cg} and Y_{cg}

PARAM,GRDPNT,n

Caratteristiche di massa calcolate rispetto a:

n=0 origine riferimento base

n>0 identificativo grid point

$$[W] = \begin{bmatrix} W_x & W_{12} & W_{13} & W_{14} & W_{15} & W_{16} \\ W_{21} & W_y & W_{23} & W_{24} & W_{25} & W_{26} \\ W_{31} & W_{32} & W_z & W_{34} & W_{35} & W_{36} \\ W_{41} & W_{42} & W_{43} & I_x & W_{45} & W_{46} \\ W_{51} & W_{52} & W_{53} & W_{54} & I_y & W_{56} \\ W_{61} & W_{62} & W_{63} & W_{64} & W_{65} & I_z \end{bmatrix}_{6\times6}$$

```
ID MSC, MCSO42.DAT $
TIME   10       $
SOL 200          $    OPTIMIZATION
.
.
$
DRESP1          41            WT33          WEIGHT        3             3
DRESP1          42            WT26          WEIGHT        2             6
DRESP1          43            WT34          WEIGHT        3             4
$
$  THE CENTER OF GRAVITY (CG)
$
DRESP2          50            CG-X          200
                DRESP1        41            42
DRESP2          51            CG-Y          200
                DRESP1        41            43
DEQATN          200           F(R1,R2)  =   R2/R1
$
ENDDATA
```

Structural Design Optimization
General Features

- **Solution Sequence**
  - SOL 200
- **Analysis Types supported**
  - Statics
  - Normal Modes
  - Buckling
  - Direct Frequency Response
  - Modal Frequency Response
  - Modal Transient Response
  - Static Aeroelastic
  - Aeroelastic flutter
  - Direct and Modal Complex Eigenvalue

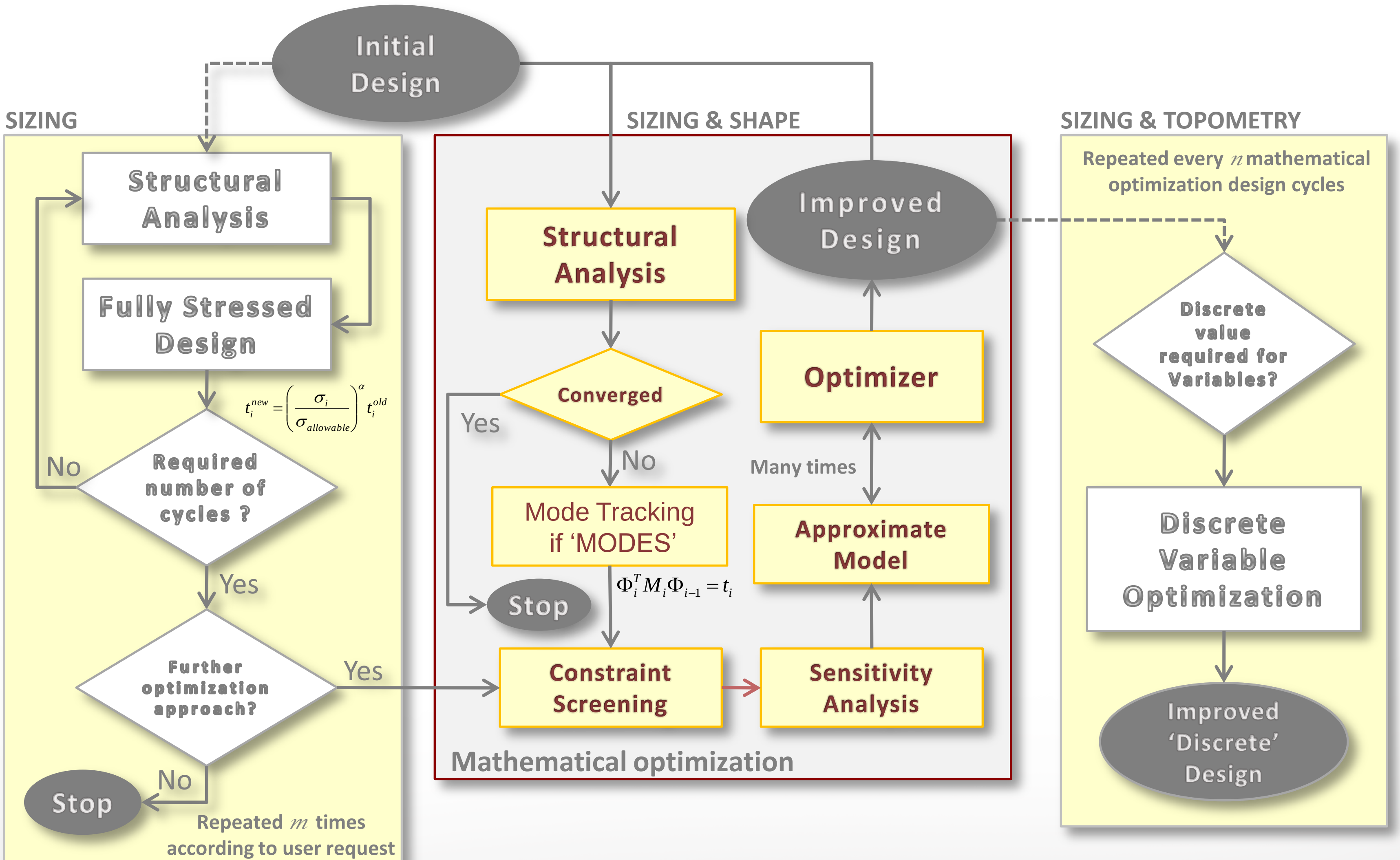
Internal and External Acoustic Analysis Responses can be considered

Multidisciplinary Solution

- **Permanent Glued Contact is supported in any of the optimization classes**

MSC Nastran - Structural Optimization

MSC Nastran flow optimization logics

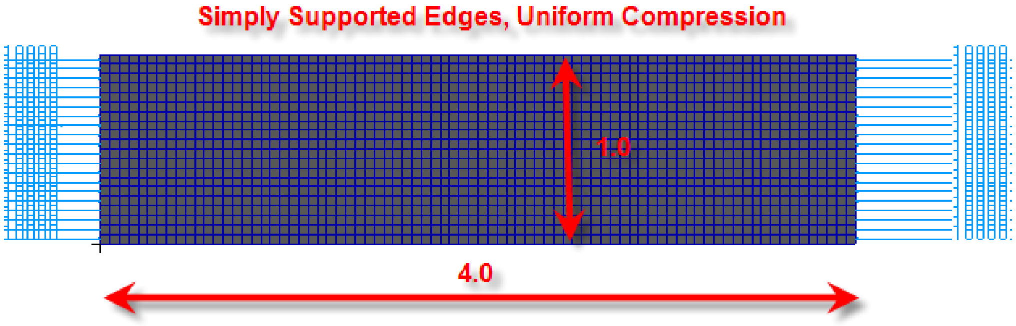


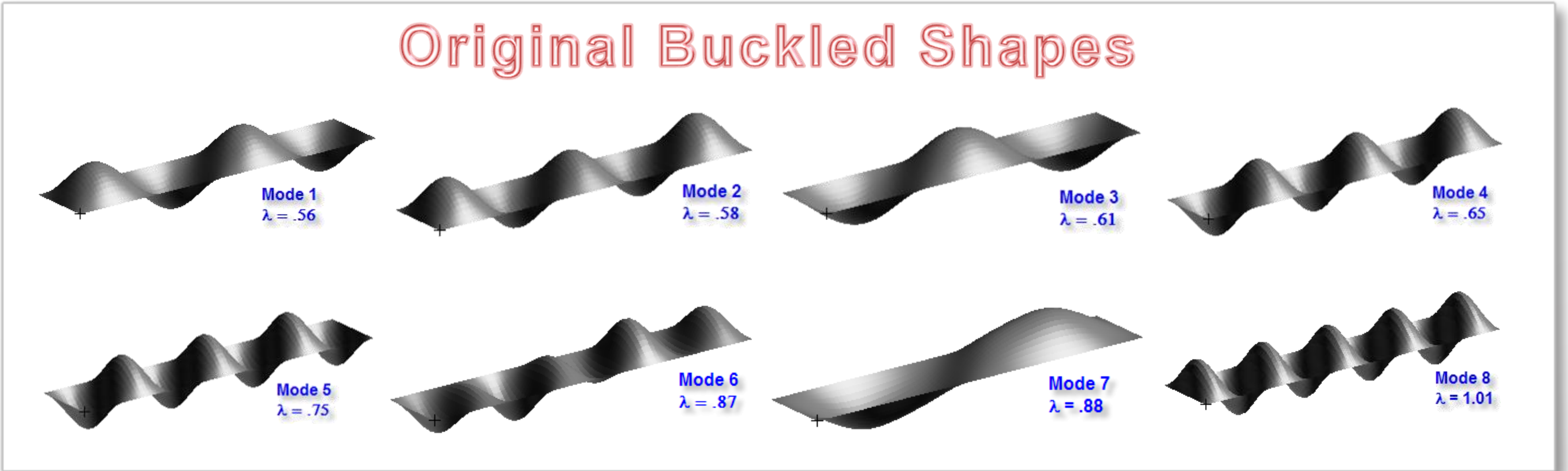
Topometry Optimization

- **Topometry Optimization is an advanced form of sizing optimization in which the element by element distribution of sizing dimensions over the structure is optimized.**
  - This is almost equivalent to having an automatic way to perform element by element design without user input of a design variable and property relation pair for each element
  - Elegantly achieved through entry which selects a designable region and property to be designed
  - Initial, lower and upper bound of designed property value are also specified
- **One design variable is automatically generated for each element referencing the property**

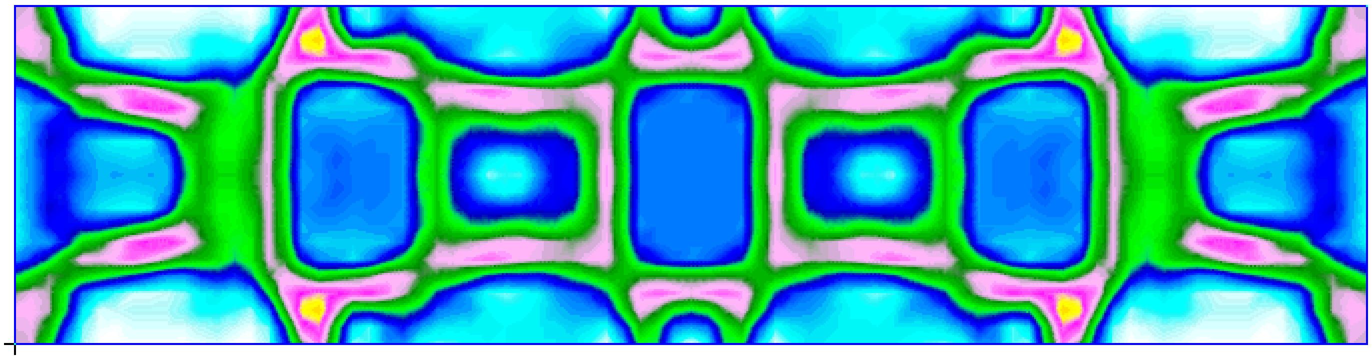
MSC Nastran - Structural Optimization
Example of Topology Optimization

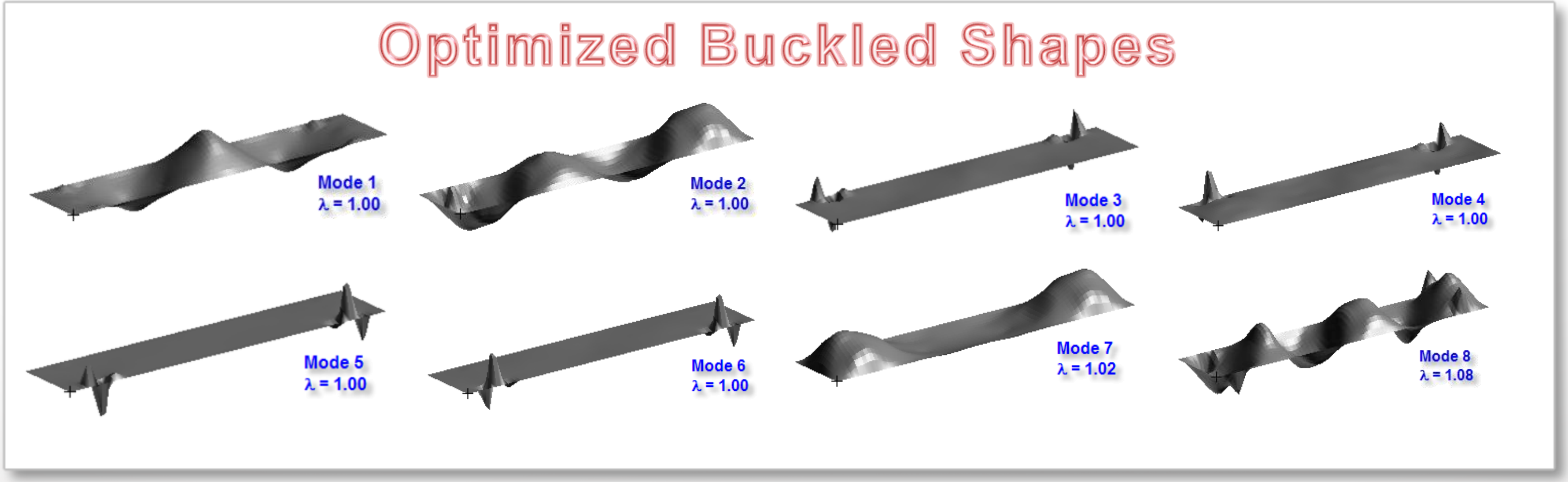
- **Topometry**
  - Material Distribution
    - By thickness/area change
  - Special form of Sizing
- **Sample Buckling**
  - Minimize Weight
  - Buckling Eigenvalue > 1.0





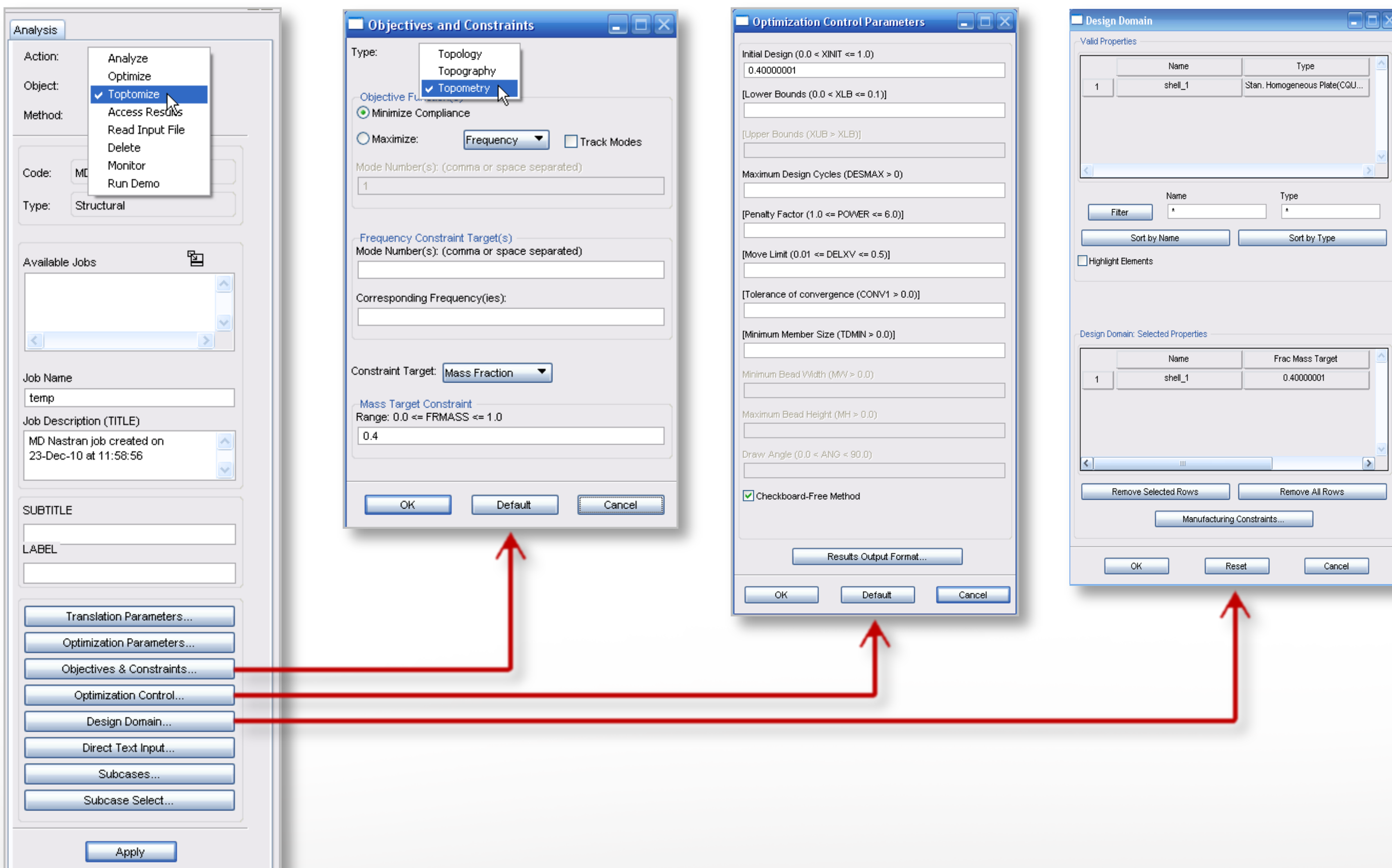
- **Progressive Material Redistribution**





MSC Nastran - Structural Optimization
Topometry - GUI

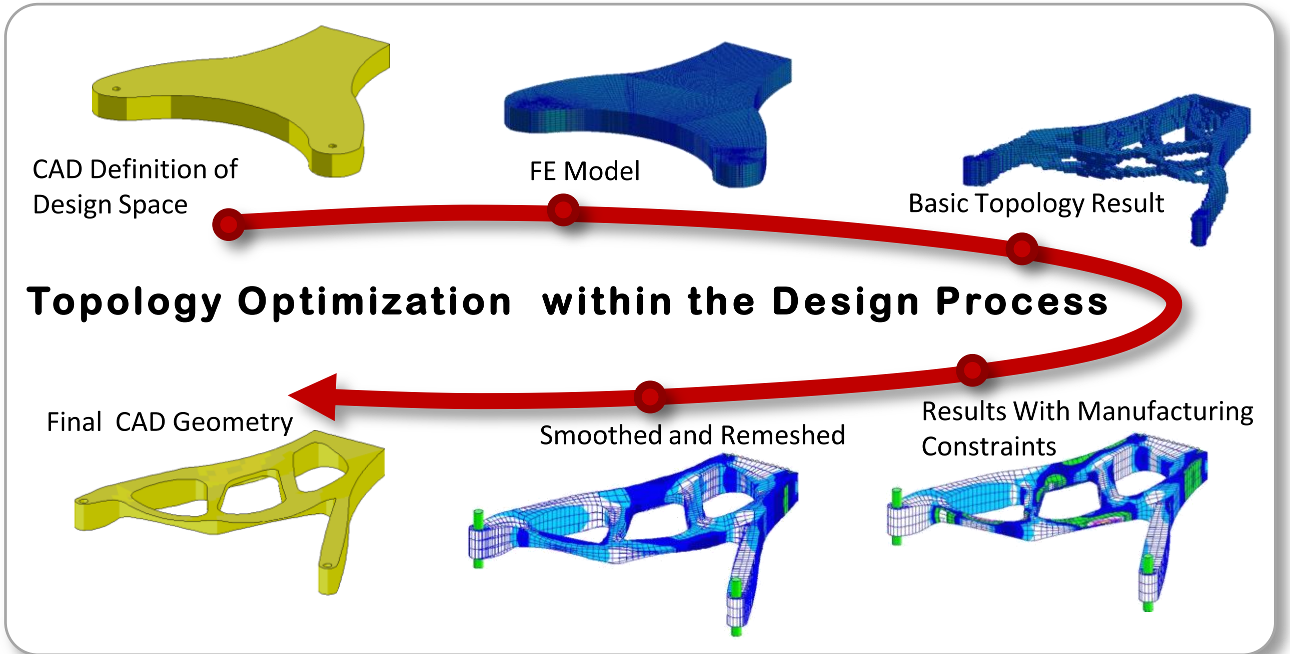
- **Topometry Optimization**
  - For ‘Toptimization’ (topometry, topology, and topography) Patran has a streamlined setup process accessed directly within the ‘Analysis’ form:



Topology Optimization

Topology Optimization
Brief Introduction

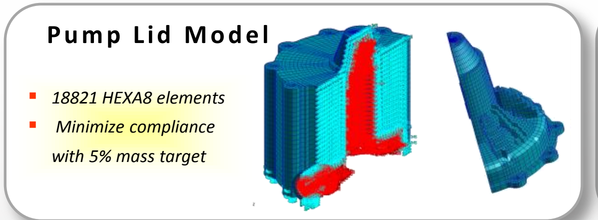
- **Topology optimization determines the optimal shape of a part**
- **The design variables are the effectiveness of each element**

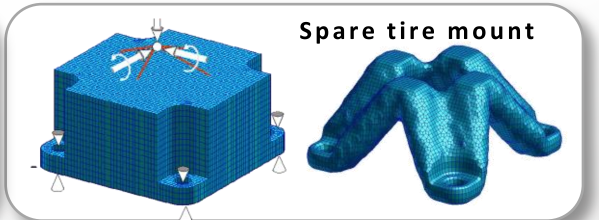


What Can I Use Topology Optimization For?

- **Static Load Path**
  - Multi-subcase
- **Frequency Constraint**
- **Frequency Response**
- **Multidisciplinary**
  - Static + Modes + Freq Response + sizing

- **Benefits**
  - Used in early design to obtain component designs and shapes
  - Used to redesign existing components





Topology Optimization

Topology Optimization Concepts

- **Density Method Approach is used in MSC.Nastran for topology optimization**
  - Alternatively called the Power Law approach or Artificial Material approach (Bendsoe, M.P., Optimal Shape Design as a Material Distribution Problem, Structural Optimization, 1, 1989, pp. 193-202*)*
  - It is based on the idea of convexification where an artificial material is used which is homogeneous.
    - The density of the artificial material can vary between 0 and 1
    - The generalized material parameters are simply taken to be proportional to the relative density
    - A power law is used to relate the density with the material property

- **The design variable x is normalized with respect to the nominal density and Young's modulus**

$$\rho = \rho_o \; x$$
$$E = E_o \; x^p$$

  - '*p*' is a penalty factor to enforce x to be close to 0 or 1 when p>1.0
    - Usually, $2 \le p \le 4$ (default = 3)

- **Each element is a DESVAR (density) which is "ideally" a discrete variable (0 or 1).**
- **A set of density values is obtained by optimization, which means the design domain is separated into solid and void regions**
- **This has the effect of <u>redistributing material</u> from regions which don't require material to those which require**

Topology Optimization
Performance Improvements with CASI

- **Detailed Topology Proposals need a fine FE Mesh**
  - Number of degrees of freedom increase
  - Leads to higher computation times
- **CASI Iterative Solver**
  - Provides a major speed up for large 3-D problems in static analysis
  - Benefit extended to topology optimization
  - Limited to compliance minimization problems
  - Invoked with the CASI option of the ITER card and SMETHDOD=ELEMENT

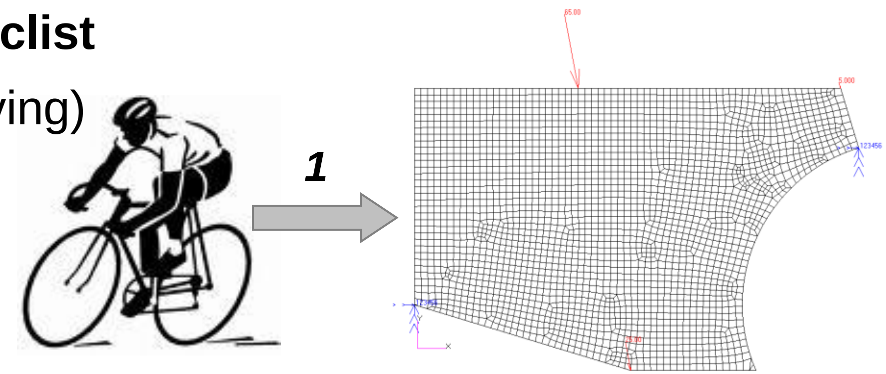
Topology Optimization
Topology Optimization Responses

- **Two response types (RTYPE) were added specifically for topology optimization**
  - COMP: compliance of structures (i.e. equivalent to total strain energy)
  - FRMASS: fractional mass. Designed mass divided by mass if all design variables are =1.0
    - Can have a separate response for each property region
- **A typical topology optimization statement:**
  - minimize the compliance (RTYPE=COMP)
  - limit the mass to a certain percentage of the maximum allowable amount (RTYPE=FRMASS)

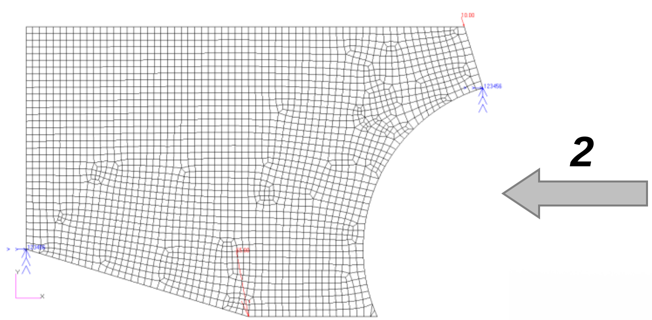
Topology Optimization

Simple example to describe process – Bicycle Frame

- **The loading conditions to be considered can be derived from the two possible positions of the cyclist**
  - Sitting (typical of normal driving)



  - Raised on the pedals (typical of a cyclist uphill or during shooting)

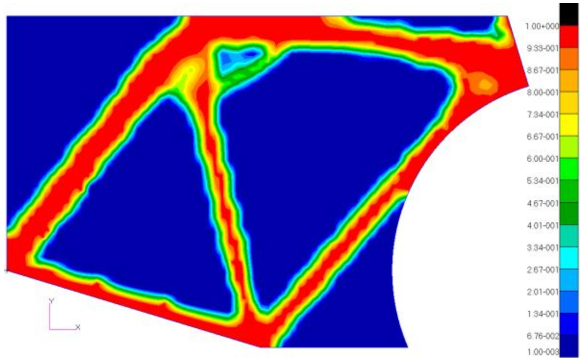


- **The objective function is defined as the sum of the values of 'compliance' associated with the above static load conditions**

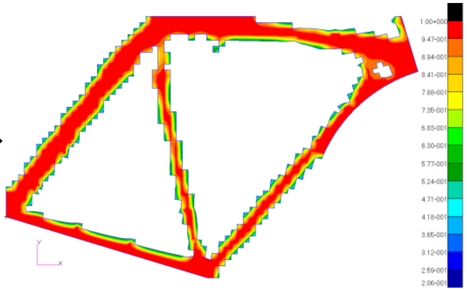
Topology Optimization

Simple example to describe process – Bicycle Frame

- **The solution obtained by topology optimization for the bicycle frame is shown**
  - Remember that the factor x (element density) varies from 0.0 to 1.0

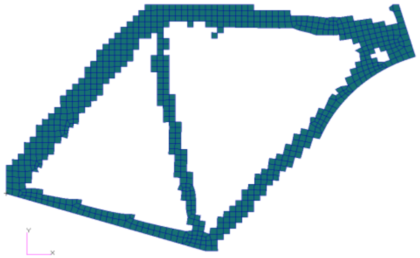


ELEMENT DENSITY DISTRIBUTION

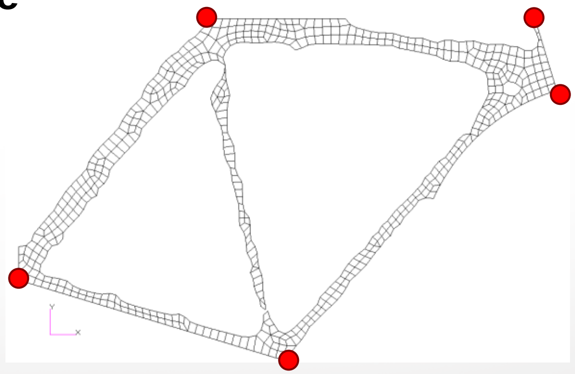


ELEMENT DENSITY DISTRIBUTION (x ≥ 0.89)

PROPOSED BYCICLE FRAME



'SMOOTHED' finite element of the frame from which the geometry passing to the CAD is generated



  - **Note how the proposed solution determine a frame that joins the five principal points of the structure defined in the design space by straight parts.**
    - Please note that these points are those in which constraints apply and loads

Topology Optimization
Manufacturability Constraints

- **Motivation for Manufacturing Constraints:**
  - Topology optimized designs may require major modifications for production or are not producible at all
- **Common issues with unrestricted topology optimization**
  - Thin beams
  - Cavities which are not achievable by casting or machining process
  - Tapered sections
  - Unsymmetric design even when loads, boundary conditions and design are fully symmetric
  - Etc.
- **Minimum Member Size**
  - To control the size of members in a topology optimal design
  - Enhances simplicity of design, hence its manufacturability
- **Casting Constraints (Draw Direction)**
  - To prevent hollow profiles
- **Extrusion Constraints**
  - Constant cross-section along a given direction
  - Essential for designs manufactured by a extrusion process
- **Mirror Symmetry Constraints**
  - Symmetry Constraints force a symmetric design in all cases
  - Support regular or irregular mesh

Topology Optimization
Design Variables for Topology

- **To select a topologically designable region, the user needs to specify a group of elements**
  - All elements referencing a given property ID are made topologically designable with the Bulk Data entry **TOPVAR** referencing that property ID.
- **Topology design variables are automatically generated with one design variable per designable element.**
- **The manufacturability constraints are then applied on all elements referencing the given property ID**

1	2	3	4	5	6	7	8	9	10
TOPVAR	ID	LABEL	PTYPE	XINIT	XLB	DELXV	POWER	PID	
	"SYM"	CID	MSi	MSi	MSi	CS	NCS		
	"CAST"	CID	DDi	DIE					
	"EXT"	CID	EDi						
	"TDMIN"	TV							

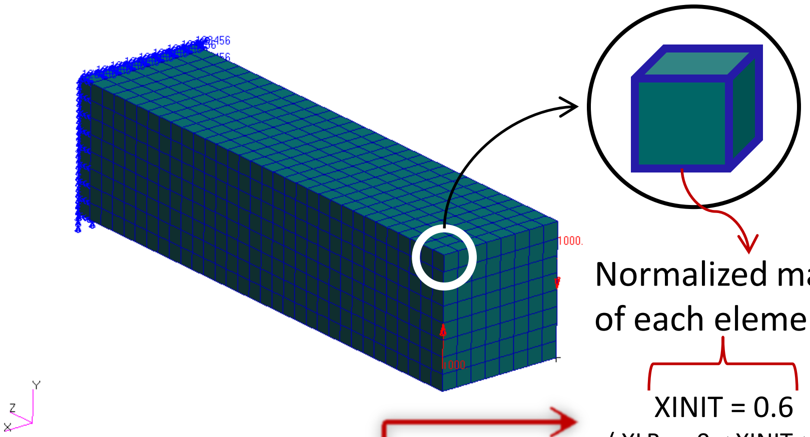
1	2	3	4	5	6	7	8	9	10
TOPVAR	2	PS1	PSOLID	0.3				10	
	SYM	5	XY	ZX					
	CAST	5	X	2					
	TDMIN	0.6							

Find the material distribution for the region defined by property PID=10 such that:

- *the resulting design is symmetric about xy and zx planes,*
- *the splitting plane is optimized for casting along x-direction (wrt coordinate system of ID 5).*
- *Also the minimum member size should not be lower than 0.6*

Topology Optimization

Design Variables for Topology - Symmetric constraint option



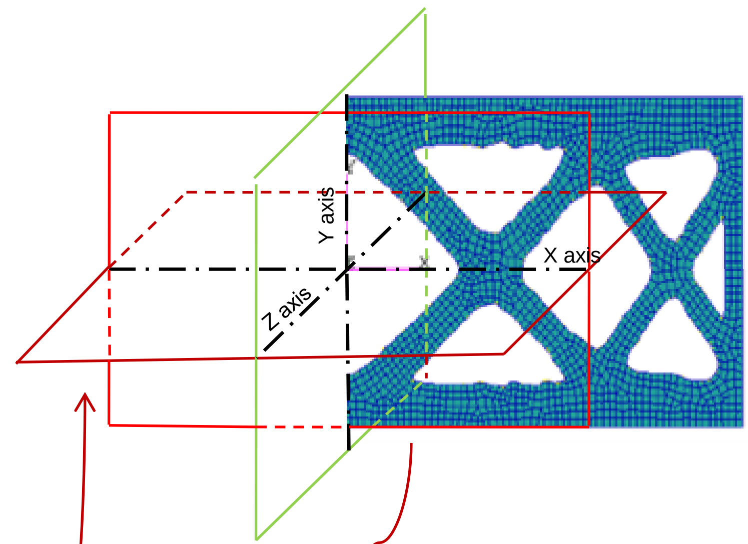
Normalized mass of each element

XINIT = 0.6

(XLB or 0 < XINIT<=1)

Lower bound of TOPVAR

TOPVAR	ID	LABEL	PTYPE	XINIT	XLB	DELXV	POWER	PID	
	"SYM"	CID	MSi	MSi	MSi	CS	NCS		
	"CAST"	CID	DDi	DIE	ALIGN				
	"EXT"	CID	EDi	ALIGN					
	"TDMIN"	TV							



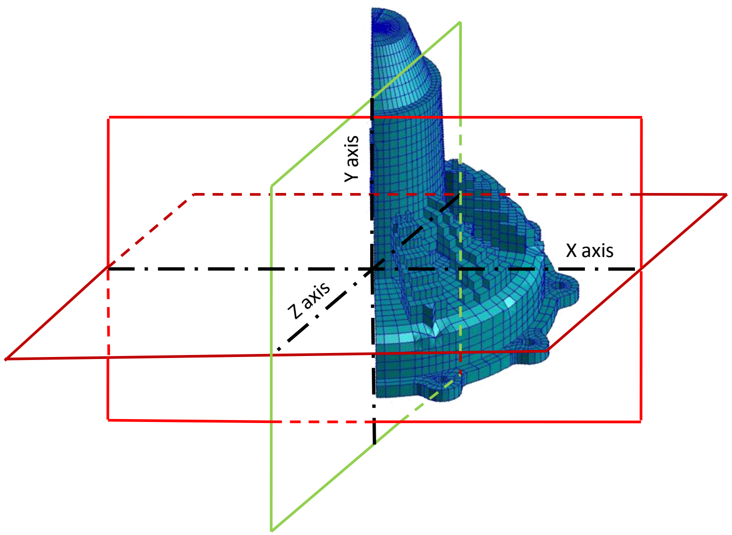
MSi = XZ (plane)

Cyclic Symmetry Option

	"SYM"	CID	MSi	MSi	MSi	CS	NCS		

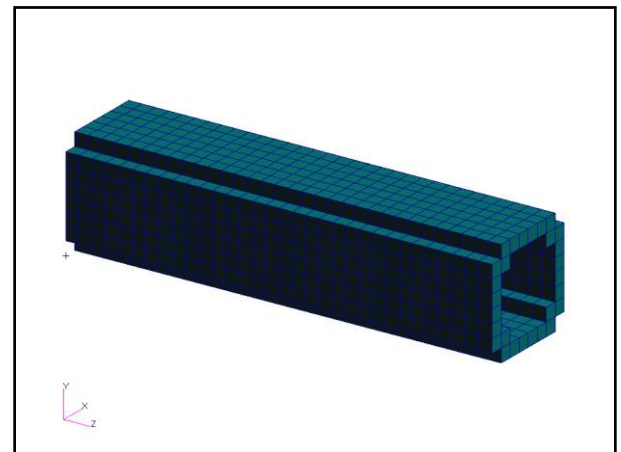
Topology Optimization

Casting and extrusion constraints



DDi (Die Draw Direction) = Y (axis)

	"CAST"	CID	DDi	DIE	ALIGN				



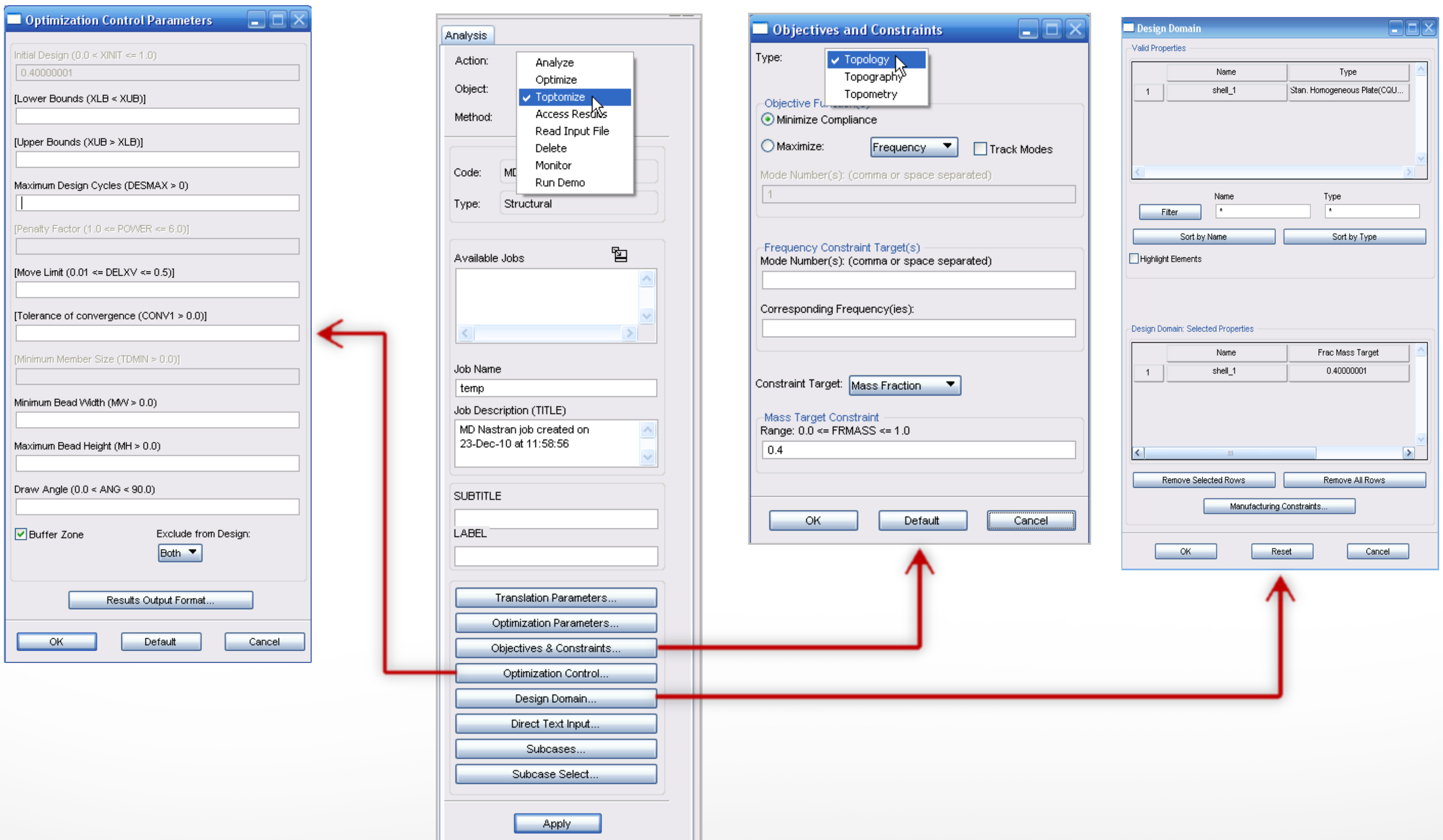
EDi (Extrusion Direction) = Z (axis)

	"EXT"	CID	EDi	ALIGN					

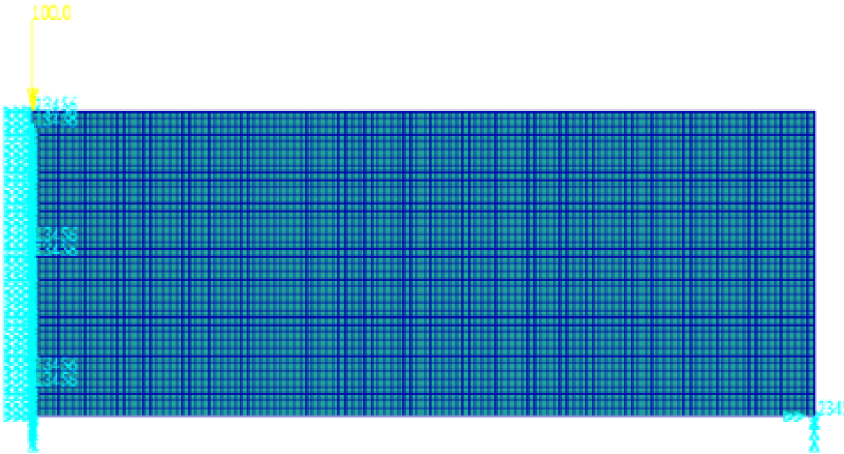
Indicates whether the designed property finite element mesh is precisely aligned with the draw direction or extrusion direction.

Topology Optimization
GUI

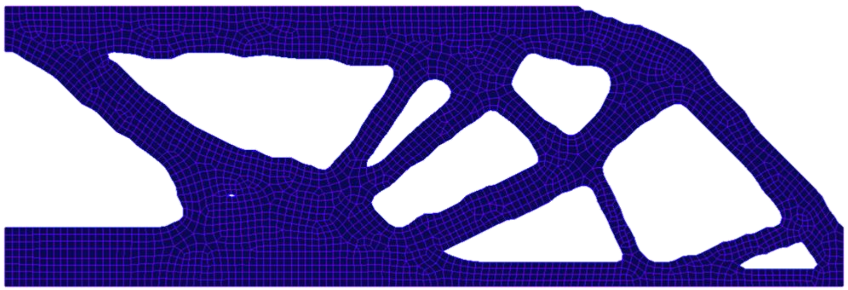
- **Similarly to topometry discussed earlier topology optimizations can be pre-processed in Patran’s streamlined setup process accessed directly within the ‘Analysis’ form:**



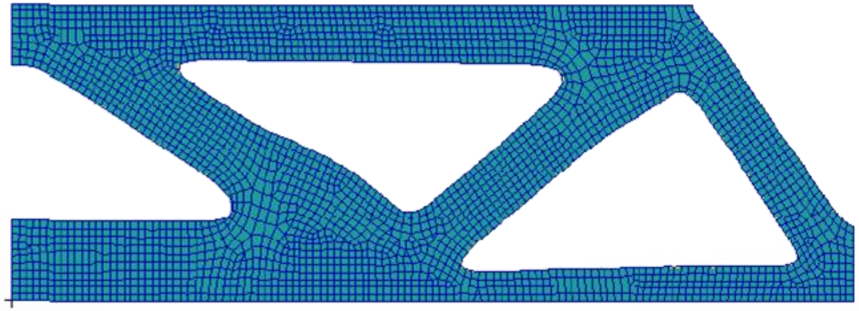
Topology Optimization
Example with Manufacturing Constraints



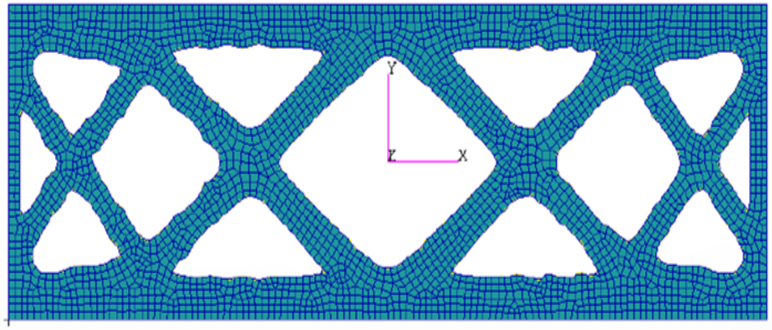
Base Model: MBB Beam



Baseline Topology - “organic design”



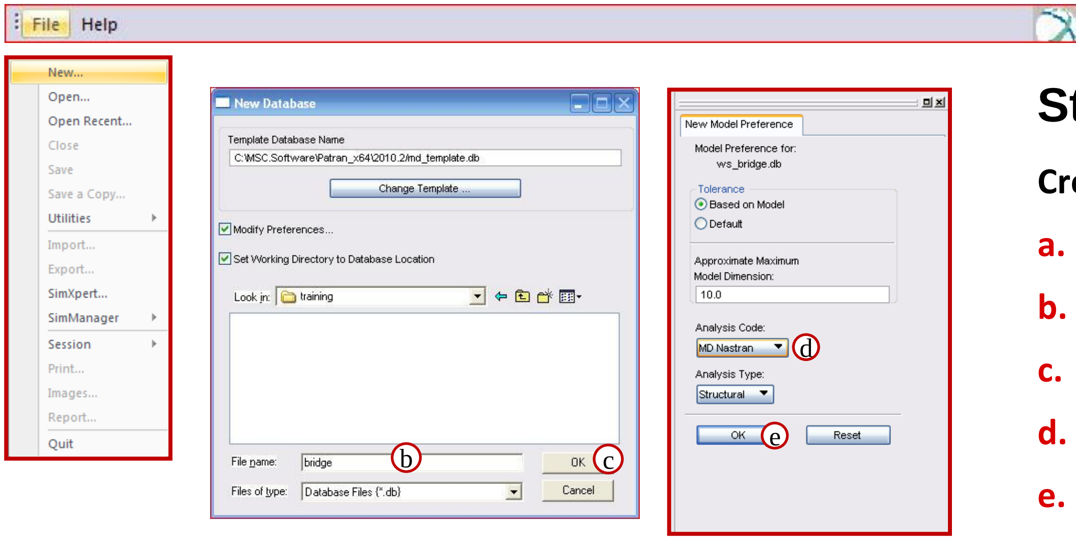
Minimum Member Size



Symmetry Constraint

Topology Optimization Workshop

Topology Optimization of a Bridge - STEP 1 and 2



Step 1. Create a New Database

Create a new database. Name it bridge.db.

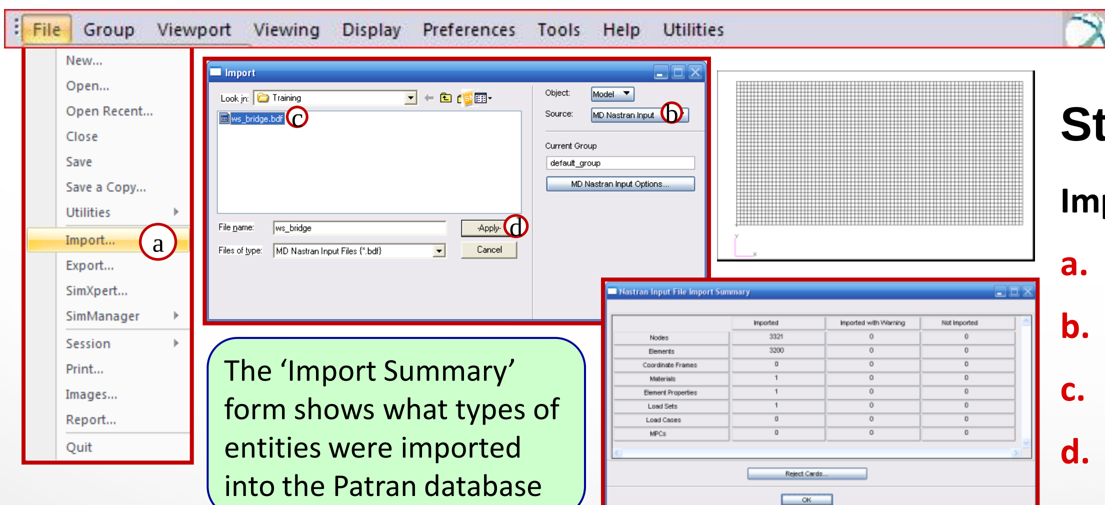
a. Pull down File > New.

b. Enter bridge as the *File name*.

c. Click OK.

d. Make sure MD Nastran is the *Analysis Code*

e. Click OK.



The 'Import Summary' form shows what types of entities were imported into the Patran database

Step 2. Import the mesh

Import the MD Nastran model.

a. Pull down File > Import.

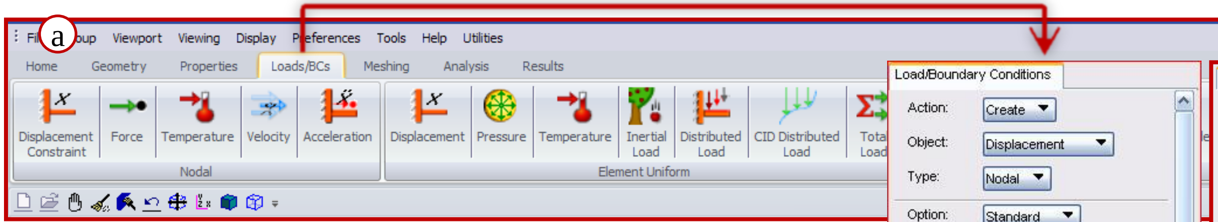
b. Select Source: MD Nastran Input.

c. Select ws_bridge.bdf as the File name.

d. Click Apply.

Topology Optimization Workshop

Topology Optimization of a Bridge – STEP 3



Constrain X and Y translations at Node 1

a. Under the *Loads/BCs* tab, click Displacement Constraint in the *Nodal* group

b. *New Set Name*: fixed12

c. Click Input Data.

d. Enter < 0,0, > for the Translations

e. Click OK

f. Click Select Application Region.

g. Pull down *Select* to FEM

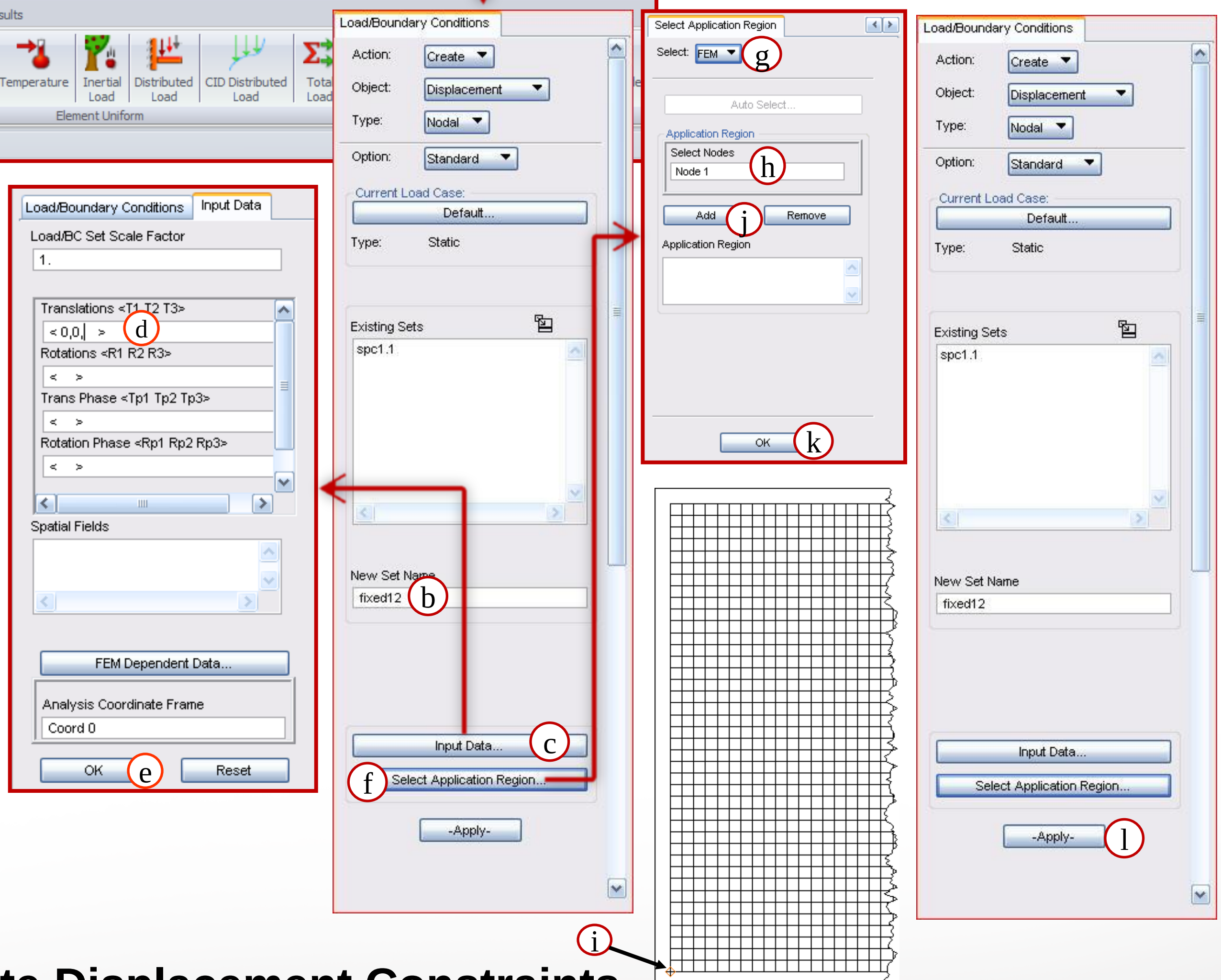
h. Click in the Select Nodes list box

i. Select the node in the lower left corner of the model (Node 1).

j. Click Add

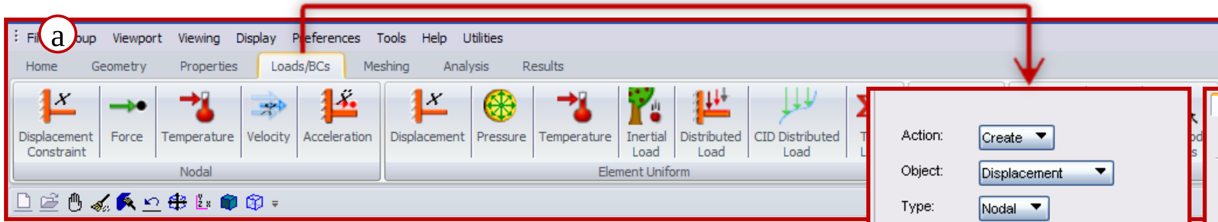
k. Click OK

l. Click Apply



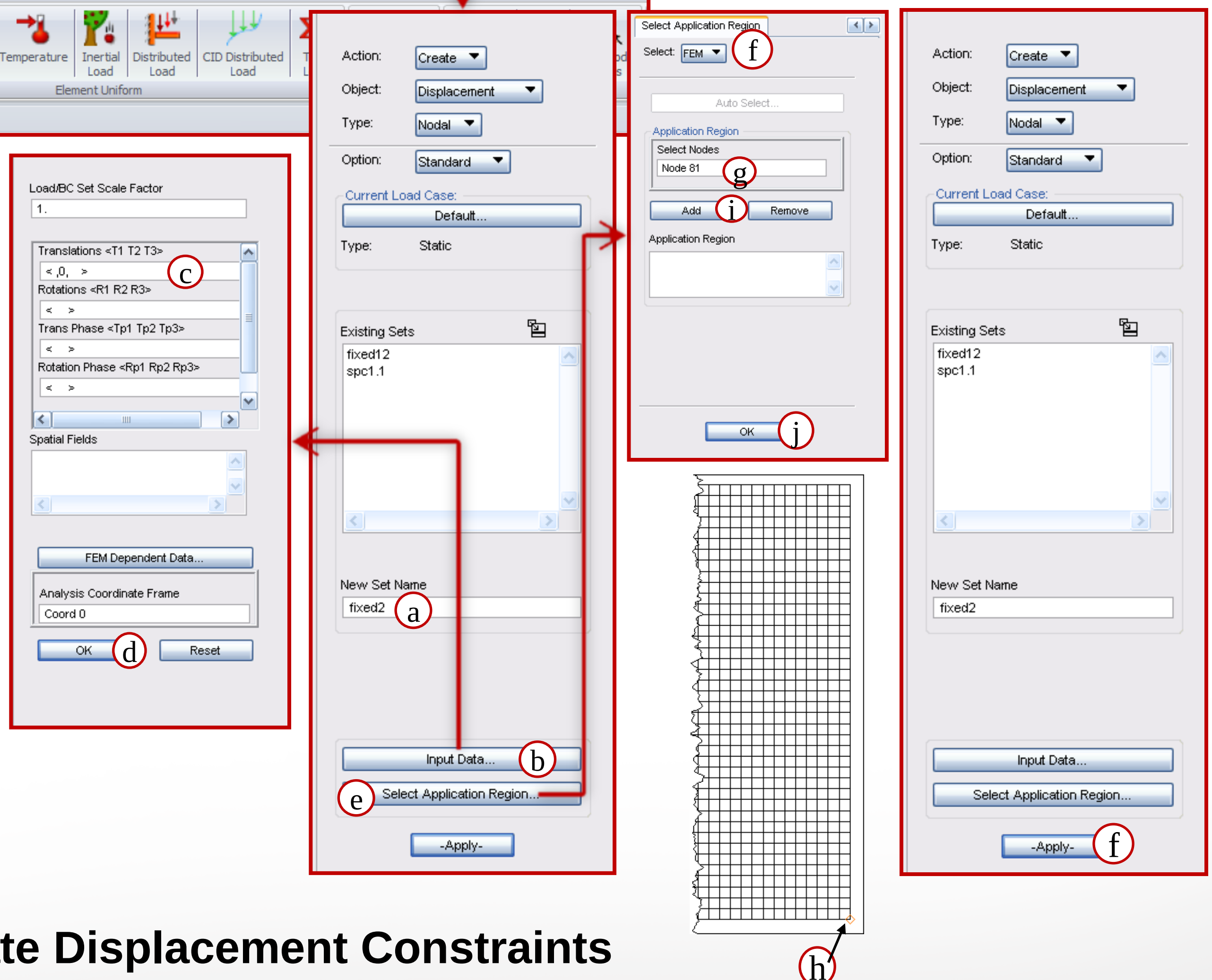
Step 3. Create Displacement Constraints

Topology Optimization of a Bridge – STEP 3 (Cont.)

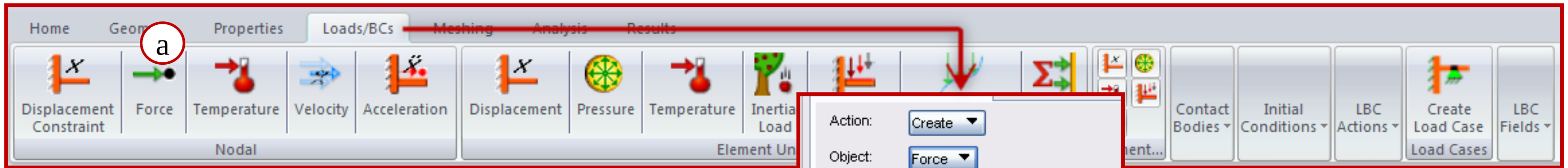


Create displacement boundary conditions for the right side.

- ***a.*** ***New Set Name*: fixed2**
- **b.** **Click Input Data.**
- **c.** **Enter < ,0, > for the Translations**
- **d.** **Click OK**
- **e.** **Click Select Application Region.**
- **f.** **Select: FEM**
- **g.** **Click in the Select Nodes list box**
- **h.** **Select the node in the lower right corner of the model (Node 81).**
- **i.** **Click Add**
- **j.** **Click OK**
- **k.** **Click Apply**

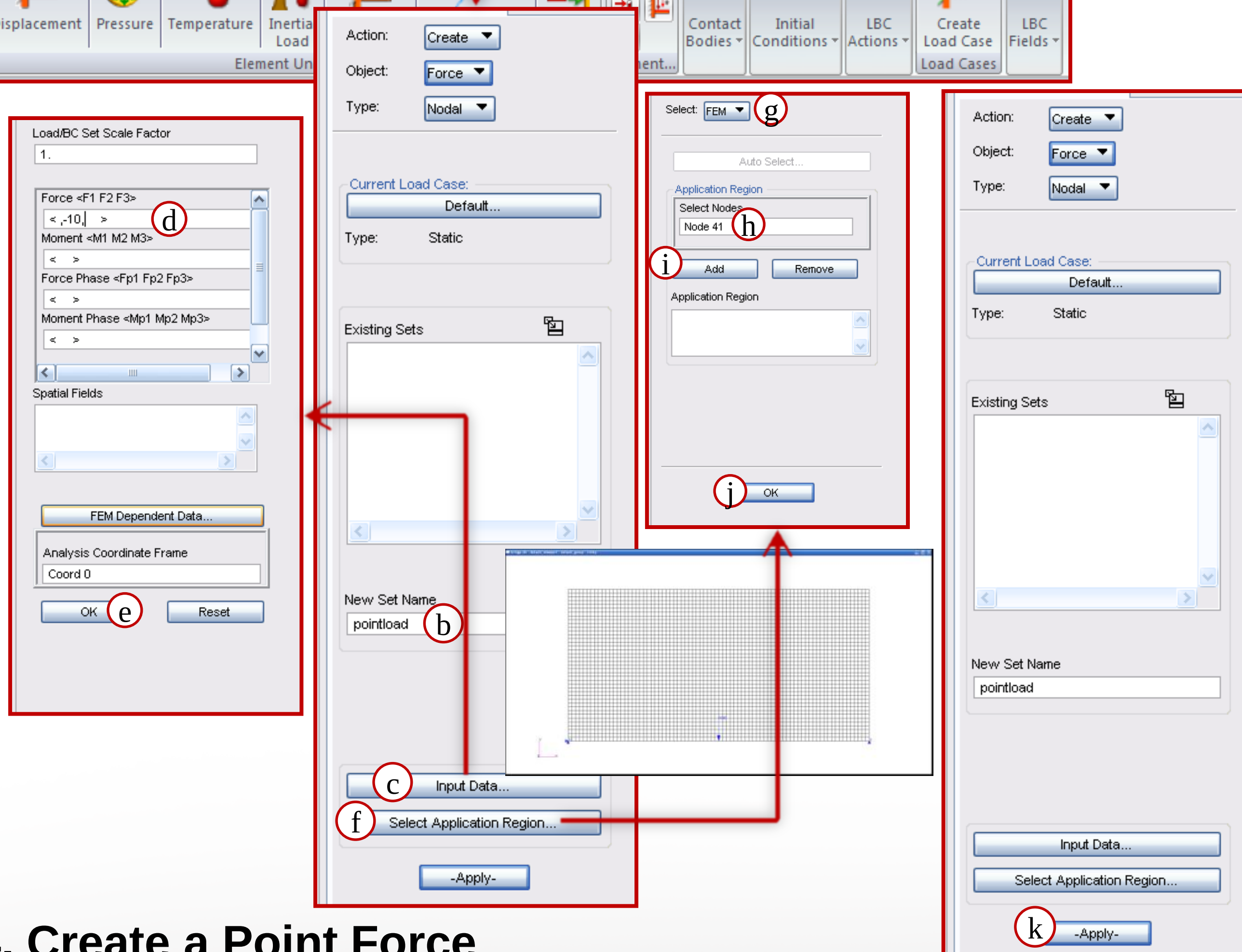


Step 3. Create Displacement Constraints



Create a point force of -10N in Y direction in the middle of the bottom edge.

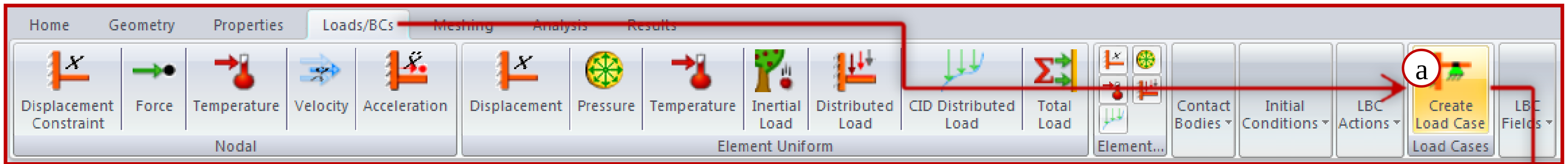
- **a.** Click Force in the *Nodal* group
- **b.** Enter pointload as *New Set Name.*
- **c.** Click Input Data
- **d.** Enter < , -10 , > as *Force* to create -10N in Y direction
- **e.** Click OK
- **f.** Click Select Application Region.
- ***g.*** *Select*: FEM
- **h.** Click in the *Select Nodes* list box and enter Node 41.
- **i.** Click Add
- **j.** Click OK
- **k.** Click Apply



Step 4. Create a Point Force

Topology Optimization Workshop

Topology Optimization of a Bridge – STEP 5



Step 5. Verify Load Case

Verify the all the loads and BC's are in the default load case

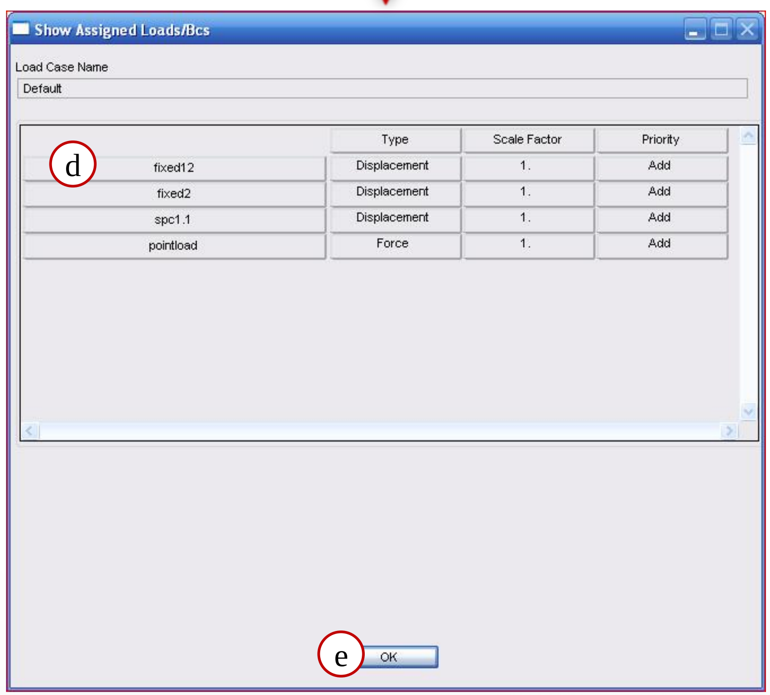
a. Click **Create Load Case** in the *Load Cases* group

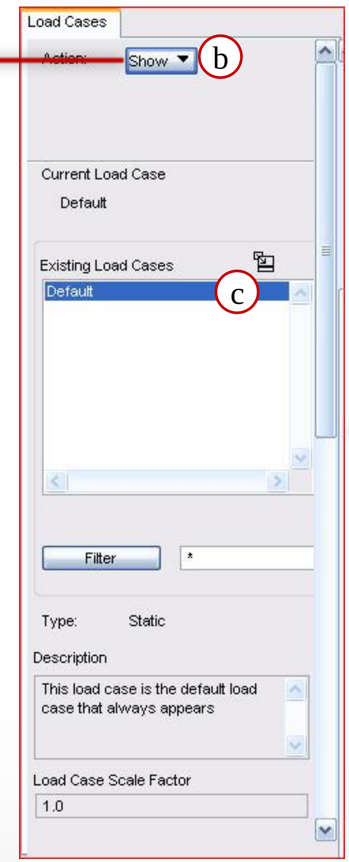
b. Pull down the *Action* to **Show**

c. Highlight **Default**

d. Verify that all 4 loads/BC are in the list of selected loads. If not ask the seminar leader!

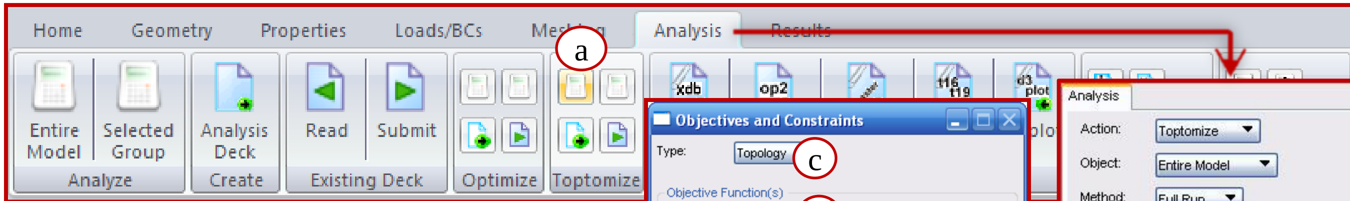
e. Click **OK**





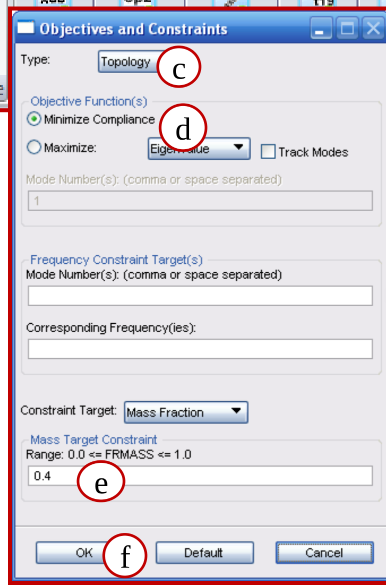
Topology Optimization Workshop

Topology Optimization of a Bridge – STEP 6

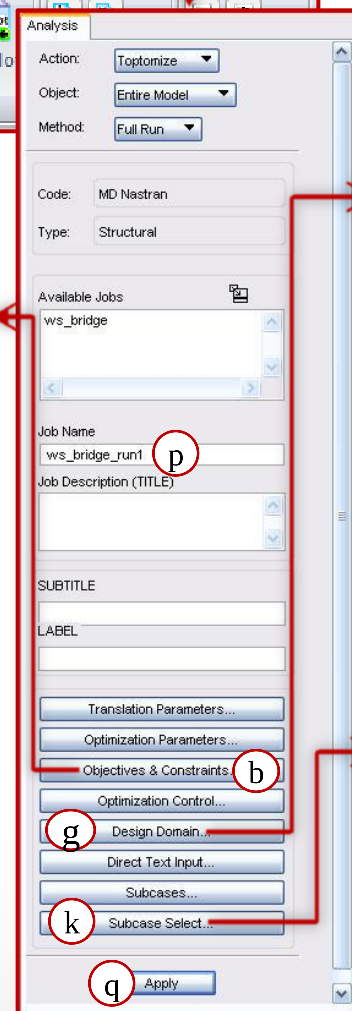


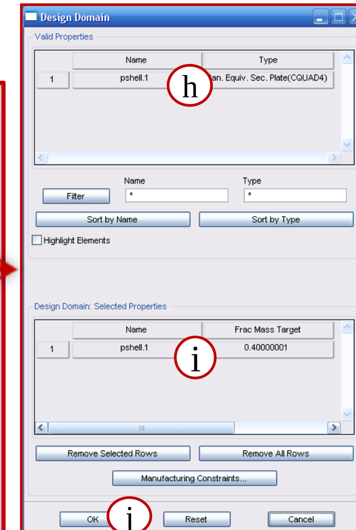
Set up a topology optimization analysis

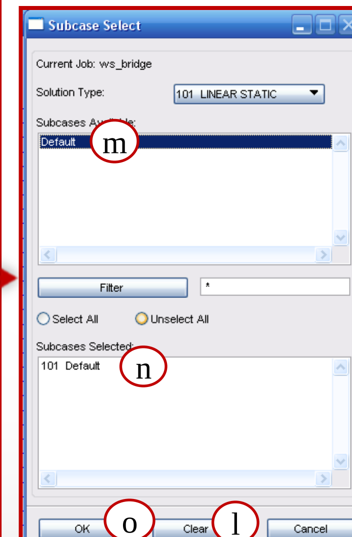
a. Under the *Analysis* tab, click **Entire Model** in the *Toptomize* group
b. Click **Objectives & Constraints**
c. Verify *Type*: **Topology**
d. Verify that **Minimize Compliance** is checked
e. Verify Constraint Target: **Mass Fraction** and target value of **0.4**
f. Click **OK**
g. Click **Design Domain**
h. Click **pshell.1** to add the property to the design domain
i. pshell.1 is now selected under **Design Domain: Selected Properties**
j. Click **OK**
k. Click **Subcase Select**
l. Click **Clear**
m. Click **Default** to select the default load case
n. It now appears under **Subcases Selected**
o. Click **OK**
p. Enter job name: **ws_bridge_run1**
q. Click **Apply**



Compliance is a measure of the flexibility of the structure and is the force vector times the displacement (F^t*u). By minimizing this, the stiffness of the structure is maximized. Mass fraction is the value of the desired remaining mass after optimization. 0.4 means that 60% of the mass will be removed



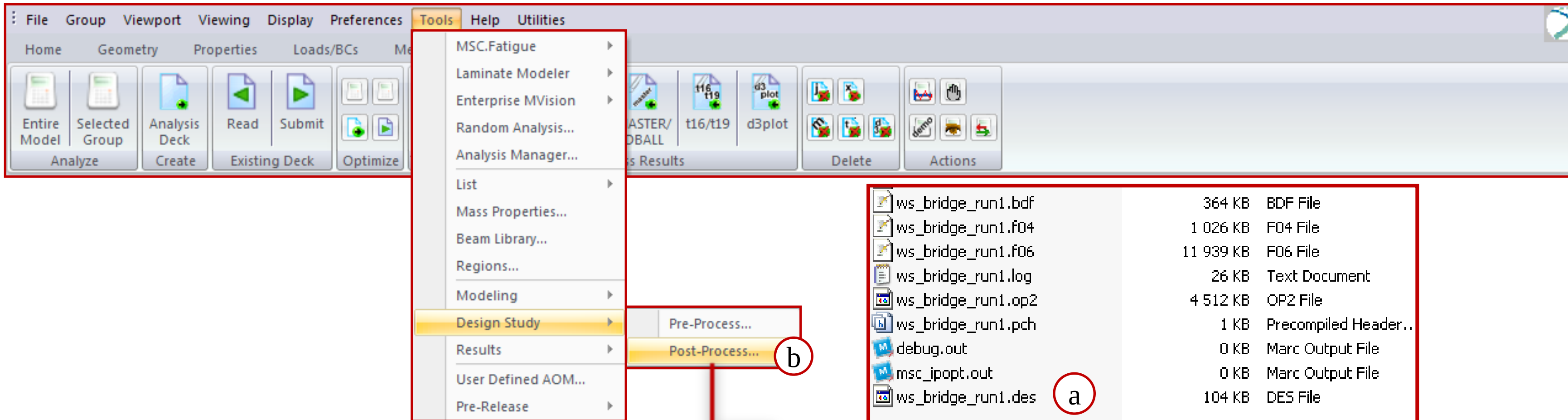




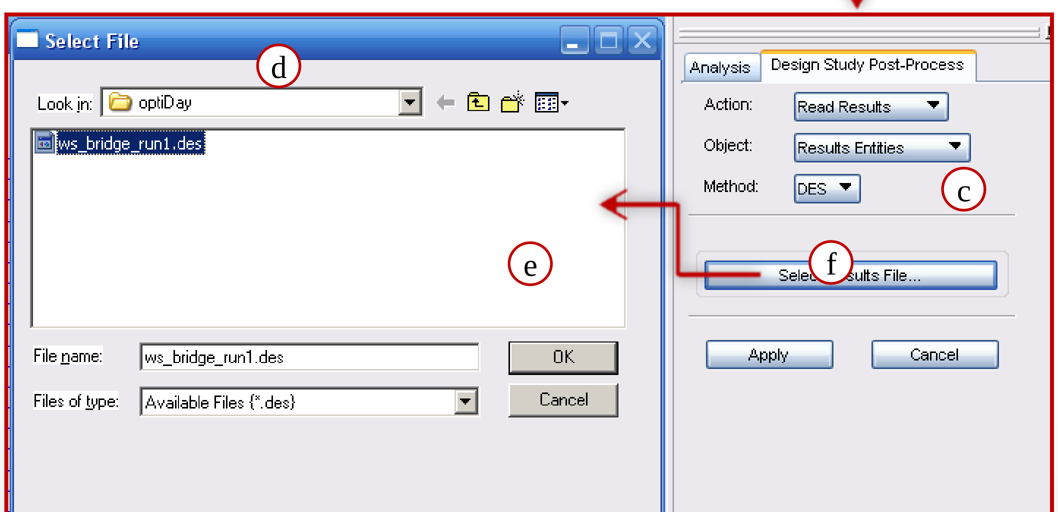
Step 6. Set up an Optimization Job

Topology Optimization Workshop

Topology Optimization of a Bridge – STEP 7



Step 7. Import Optimization Results



Import results from the Nastran analysis

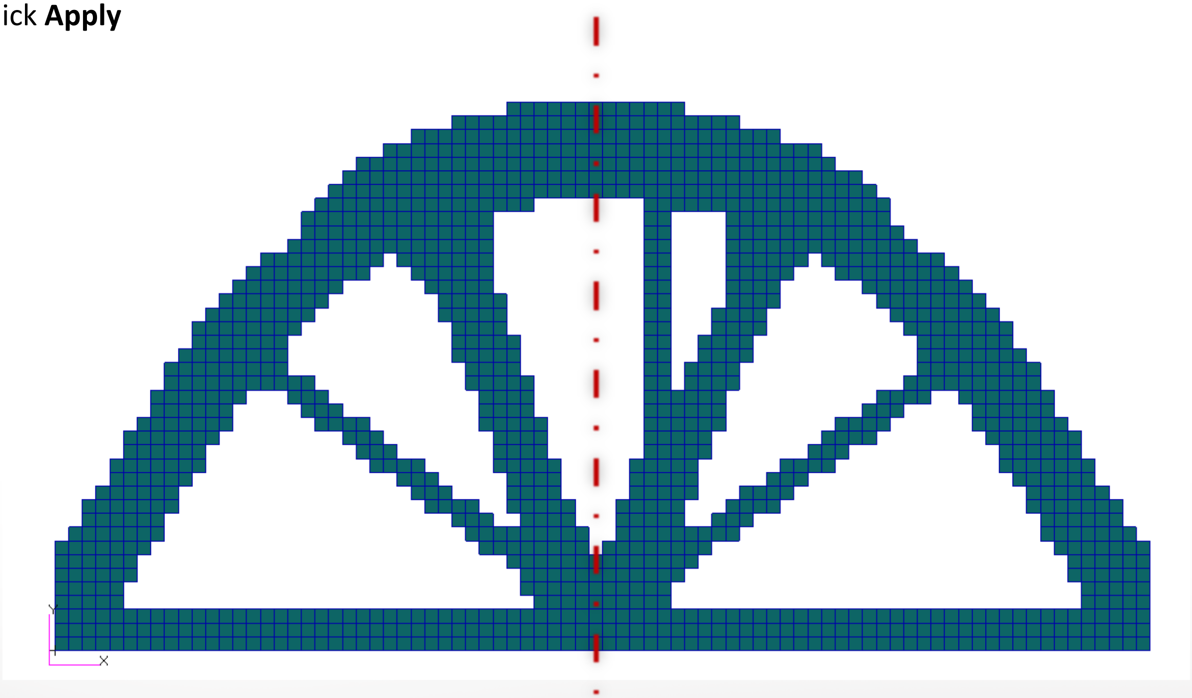
a. When you have a *.des* file in your working directory, the analysis is finished
b. In Patran, pull down **Tools > Design Study > Post-Process**
c. Click **Select Results File**
d. Select the file: **ws_bridge_run1.des**
e. Click **OK**
f. Click **Apply**

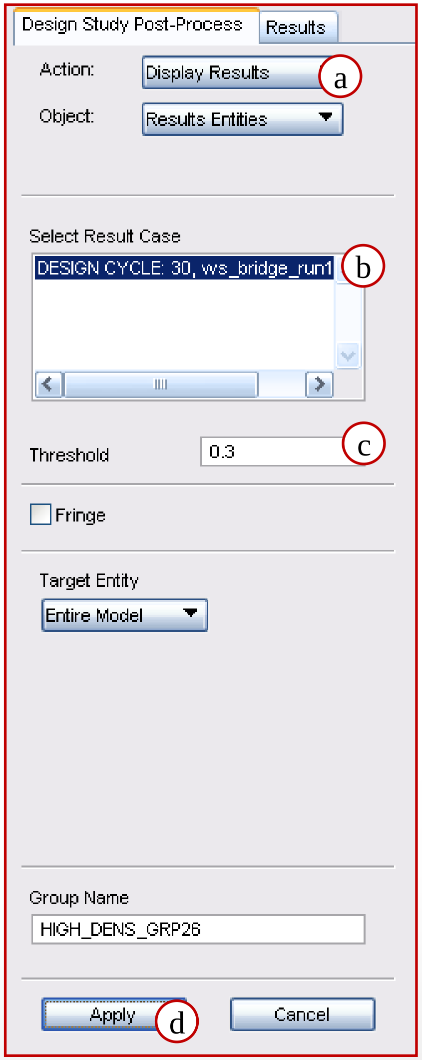
Step 8. Display Post Process Optimization Results

Post process the element density

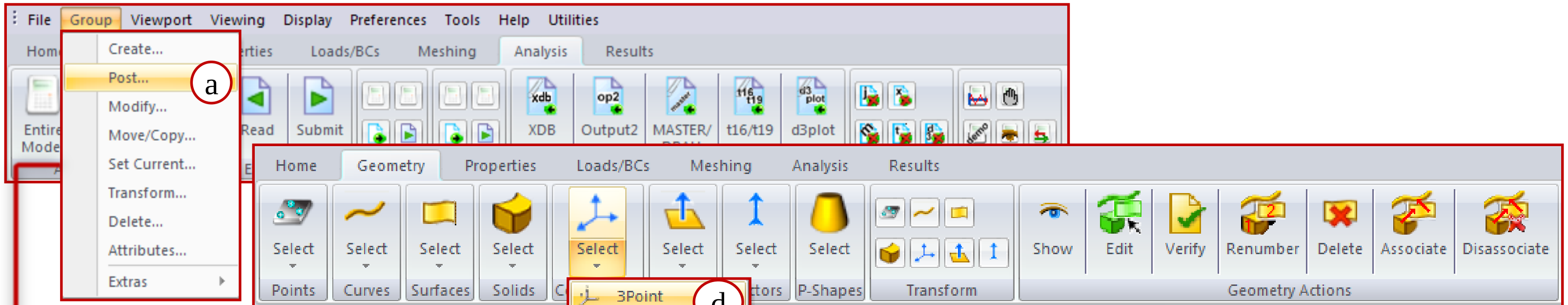
a. Pull down *Action* to **Display Results**
b. Highlight the available result
c. Set *Threshold* to **0.3**
d. Click **Apply**

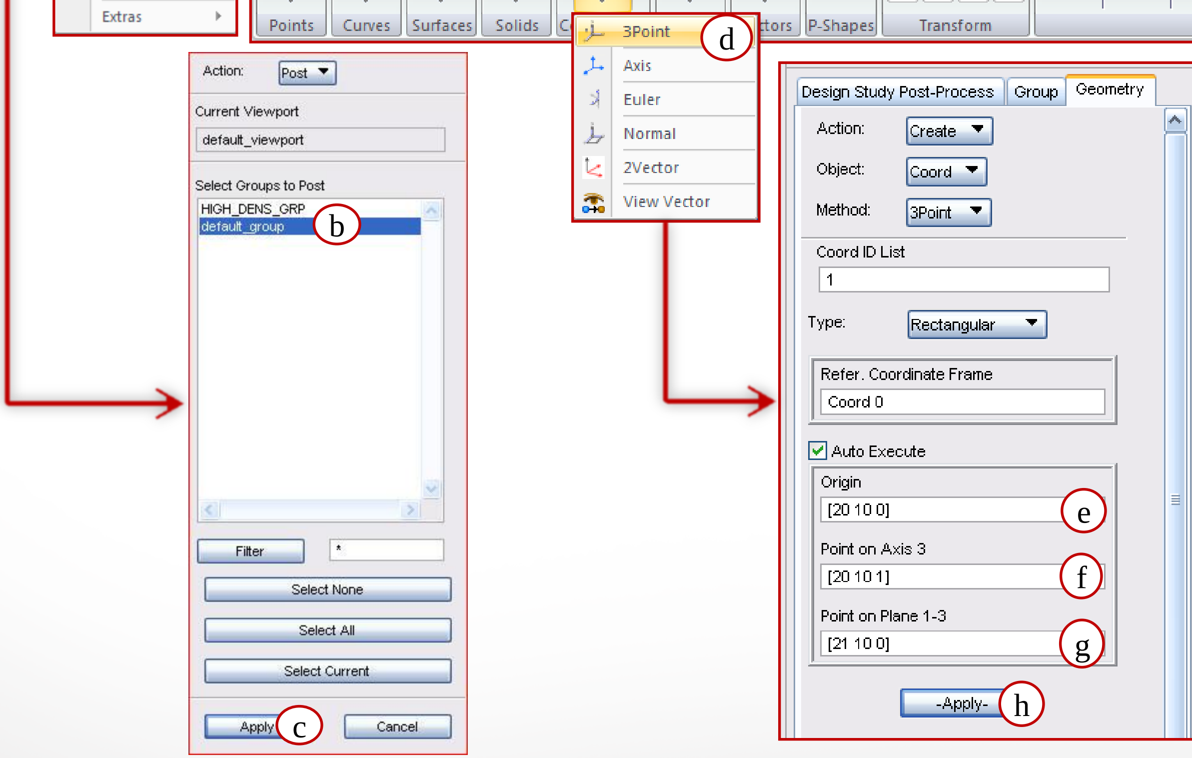
The results are not symmetric in the XZ plane. We will continue the workshop adding a coordinate system and symmetry constraints to the optimization problem





Step 9. Add a Local Coordinate System



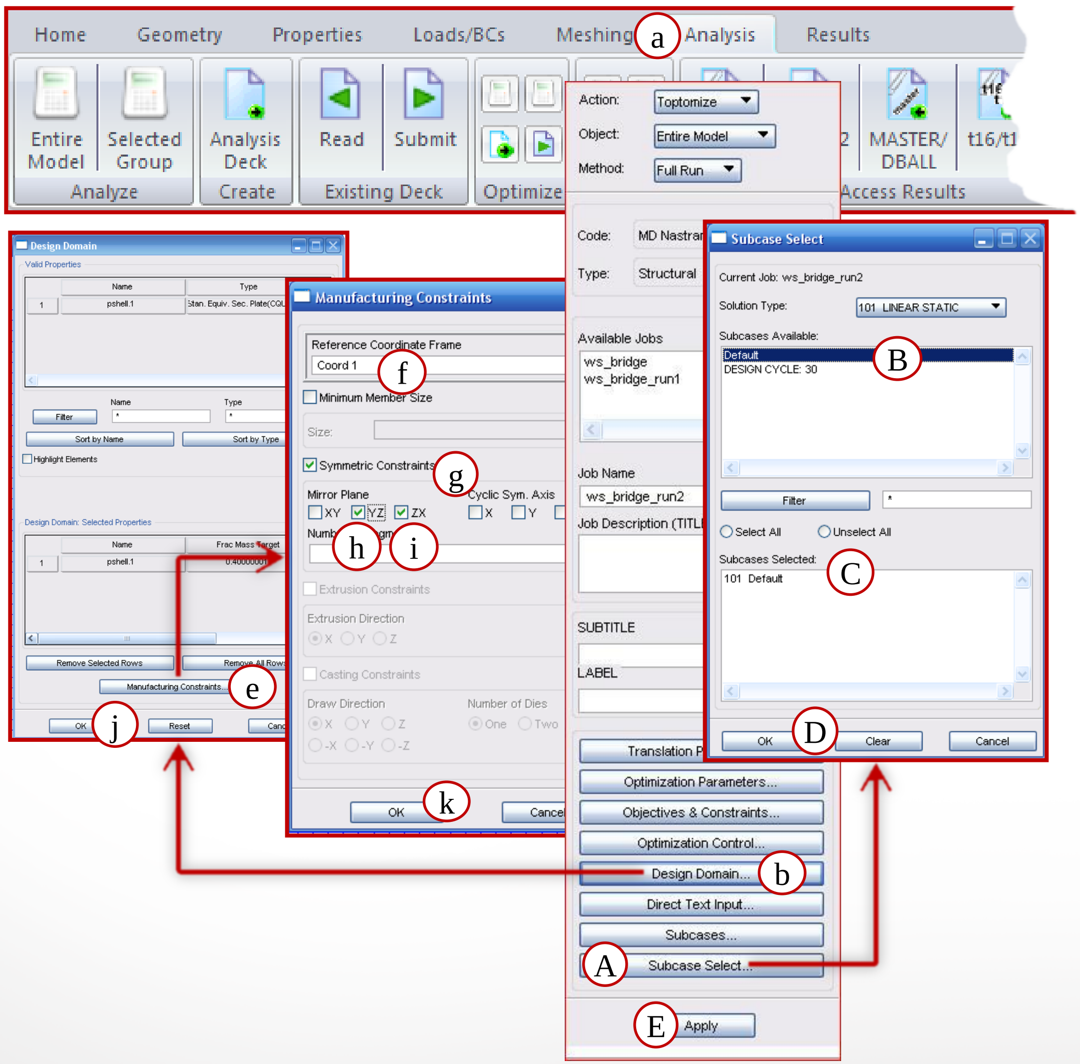


Add a local coordinate system.
(First show the complete model)

a. Pull down **Group > Post**
b. Highlight **default_group**
c. Click **Apply**
d. Under the Geometry tab, pull down **Select > 3 Point** in the *Coordinate* group
e. Enter **[20 10 0]** for *Origin*
f. Enter **[20 10 1]** for *Point on Axis 3*
g. Enter **[21 10 0]** for *Point on Plane 1-3*
h. Click **Apply**

A new coordinate system is created at the center of the model

Step 10. Create a New Job with Symmetry Constraints



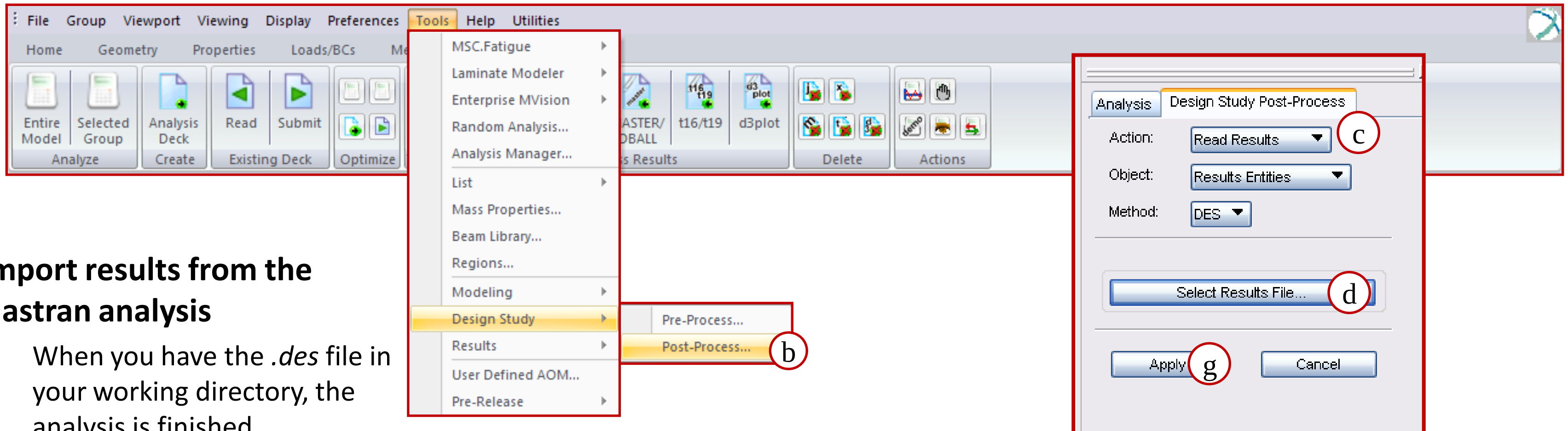
Create a new job.

a. Click the **Analysis** tab
b. Verify that the *Action* is **Toptomize**, *Object* is **Entire Model** and *Method* is **Full Run**
c. Enter **ws_bridge_run2** as Job Name
d. Click **Design Domain**
e. Click **Manufacturing Constraints**
f. Click in the *Reference Coordinate Frame* list box and select the new coordinate system by clicking on it in the viewport or entering **Coord 1**
g. Check **Symmetric Constraints**
h. Check **YZ**
i. Check **ZX**
j. Click **OK**
k. Click **OK**

Export a new Nastran input file.

A. Click **Subcase Select**
B. Click **Default**
C. Verify that only the **Default** Subcase is selected
D. Click **OK**
E. Click **Apply**

Step 11. Import Optimization Results



Import results from the Nastran analysis

a. When you have the *.des* file in your working directory, the analysis is finished

b. In Patran, pull down **Tools > Design Study > Post-Process**

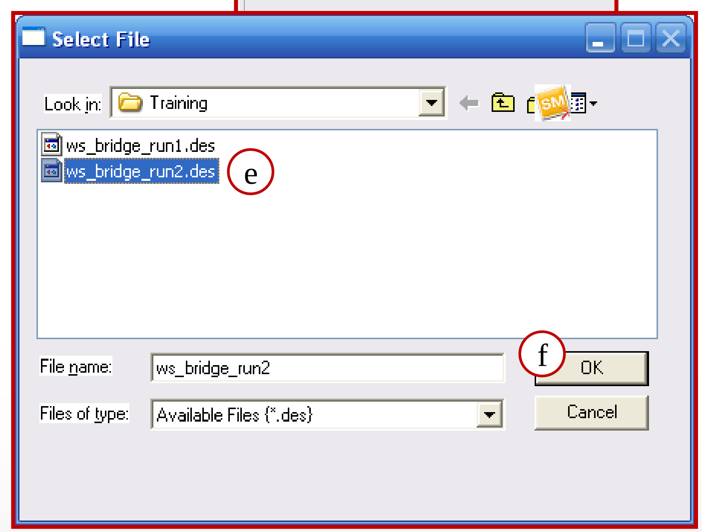
c. Pull down Action to **Read Results**

d. Click **Select Results File**

e. Select the file: **ws_bridge_run2.des**

f. Click **OK**

g. Click **Apply**



Step 12.
Display Post Process Optimization Results

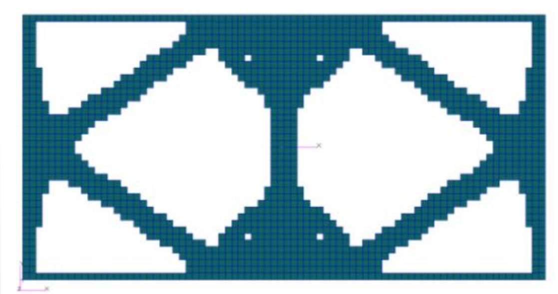
Post process the element density

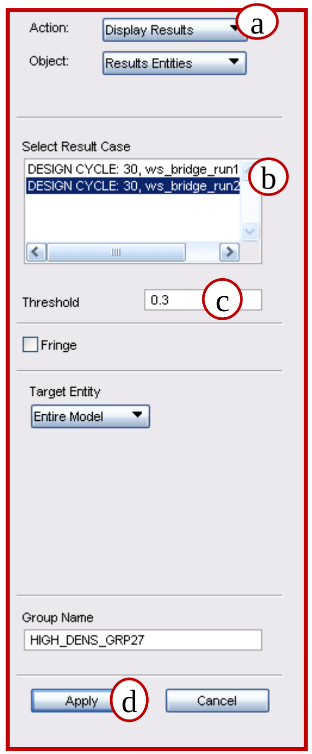
a. Pull down *Action* to **Display Results**

b. Highlight the Result Case for **ws_bridge_run2**

c. Set *Threshold* to **0.3**

d. Click **Apply**





Step 13.
Make a New Smooth Mesh

Make a new mesh from the topology optimization results

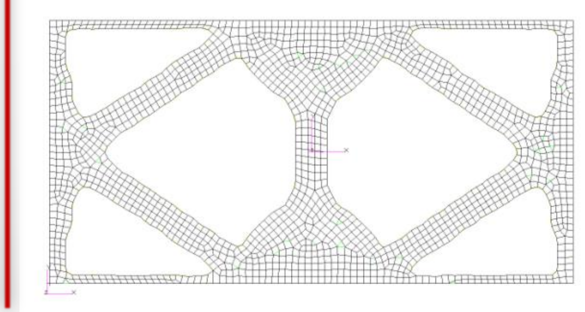
a. Pull down *Action* to **FEM Smooth**

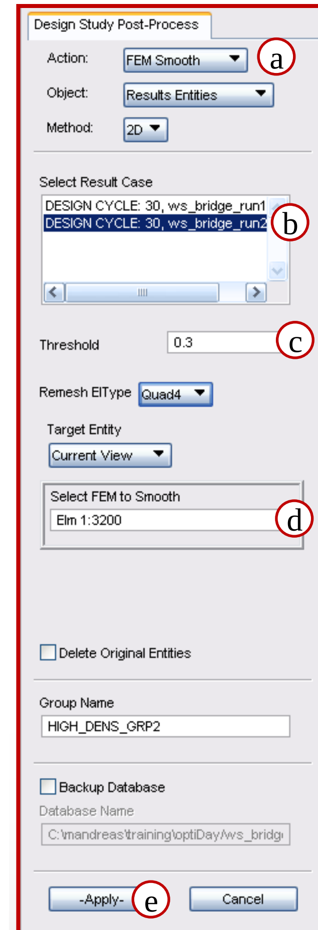
b. Highlight the Result Case for **ws_bridge_run2**

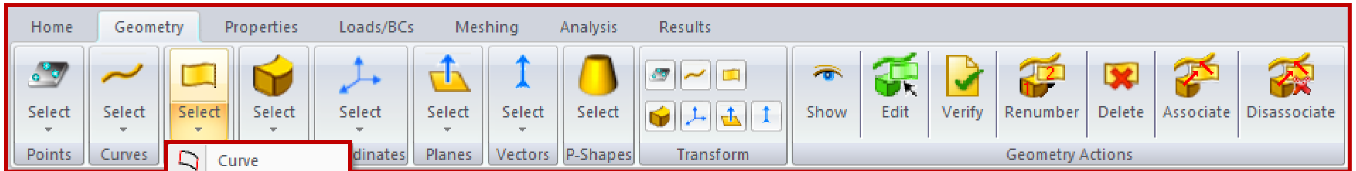
c. Set *Threshold* to **0.3**

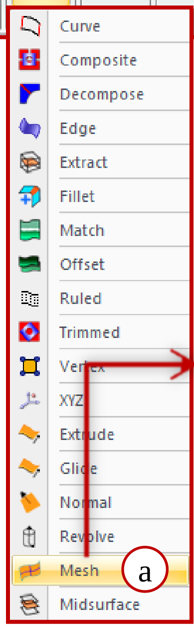
d. Select all elements with a rectangular select (hold down left mouse button and drag to make a rectangular window)

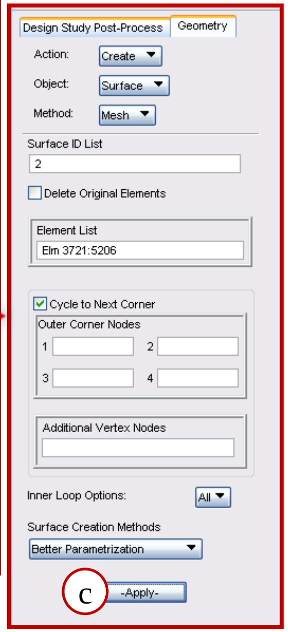
e. Click **Apply**

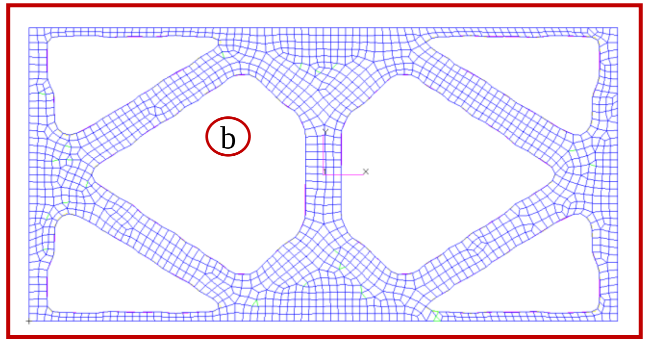










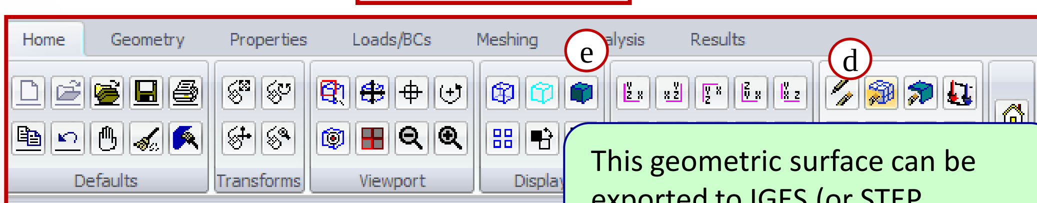


Step 14. Create a Surface

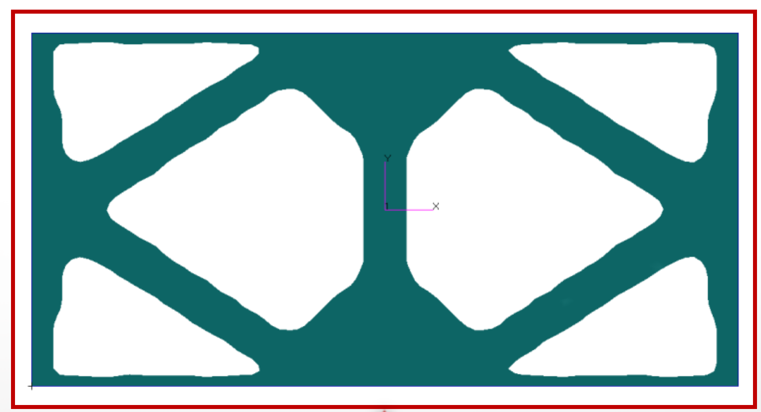
Create a surface from the mesh

a. Under the Geometry tab, pull down **Select > Mesh** in the *Surface* group
b. Drag a rectangle to select all the elements on the screen
c. Click **Apply**
d. Under the *Home* tab, click **Erase FEM** in the *Misc* group
e. Click **Smooth Shaded** in the *Display* group

Quit Patran

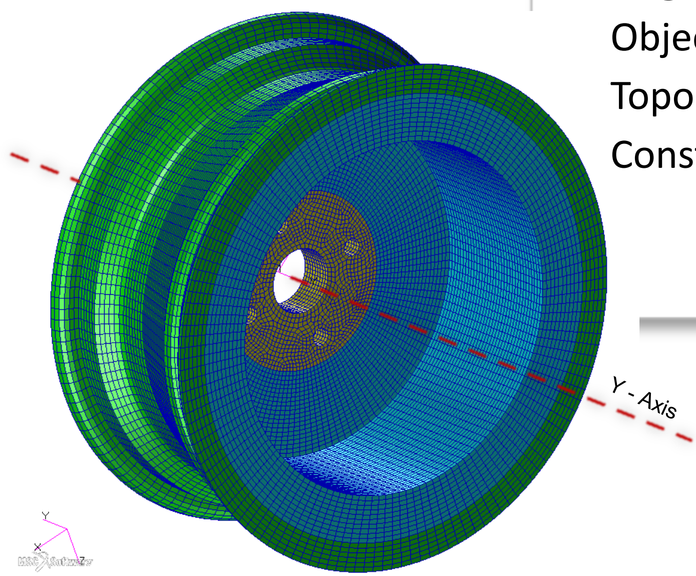


This geometric surface can be exported to IGES (or STEP, Parasolid, etc.) using **File >Export**



Topology Optimization Real Life Example

Wheel Topology Optimization



Design Model Description

Objective	Minimize averaged compliance
Topology design region	PSOLID (Blue)
Constraints	Mass Target = 0.1 (i.e. Mass saving 90%)
	The design is forced to be cyclical symmetry about Y-axis with five segments

```
DESOBJ = 10
DESGLB = 1
ANALYSIS = STATICS
SMETHOD = ELEMENT
SUBCASE 1
   SPC = 2
   LOAD = 2
BEGIN BULK
CORD2R   1          10.512  33.3312 12.9921 -22.209833.3312 4.88385
         28.388  33.3313 -19.7297
DCONSTR  1       2       .1
TOPVAR   1       PSOLID  PSOLID  .1
         SYM     1               Y               5       2
DRESP1   2       FRM     FRMASS
DRESP1   10      COMP    COMP
```

Mass Target

CS

NCS

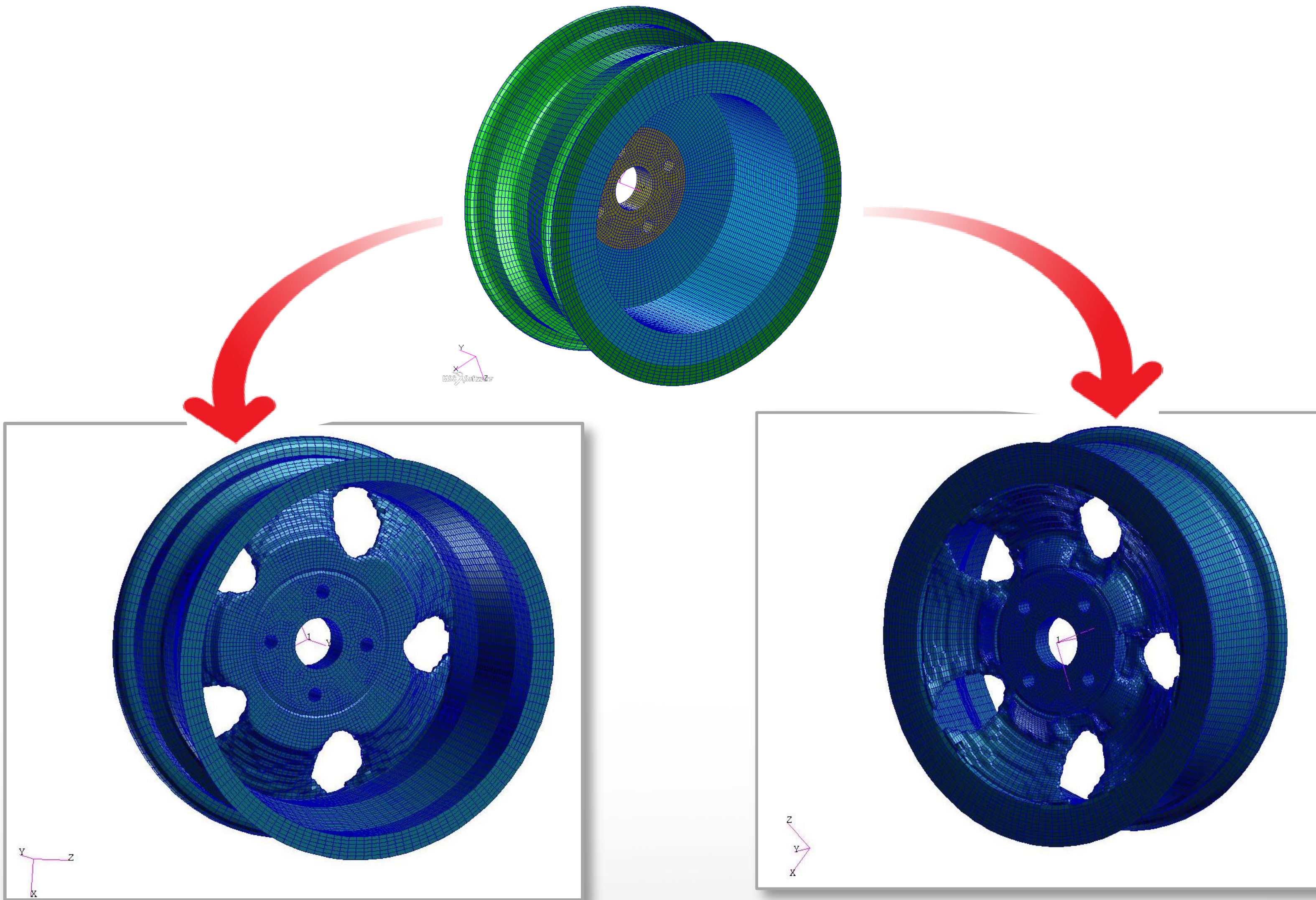
PSOLID ID

Material properties	Young's Modulus = $1.0\text{x}10^7$ Pa, Poisson's ratio = 0.3, density = 1.0
Boundary conditions	Fixed at some points
Applied loads	Force = 1000.0 N in direction of gravity
Element type	HEXA, RBE3

- **The coordinate system** (*CORD2R = 1*) **is created to be used to specify cyclical symmetry constraints**
- **The field** CS **(cyclic symmetry axis) on the SYM line is Y-axis with** NCS **(number of cyclical symmetry segments) = 5**
- **It is noticed that** *SMETHOD = ELEMENT* **is used to select CASI iterative solver**
  - Major speedup in the solution of large static analysis

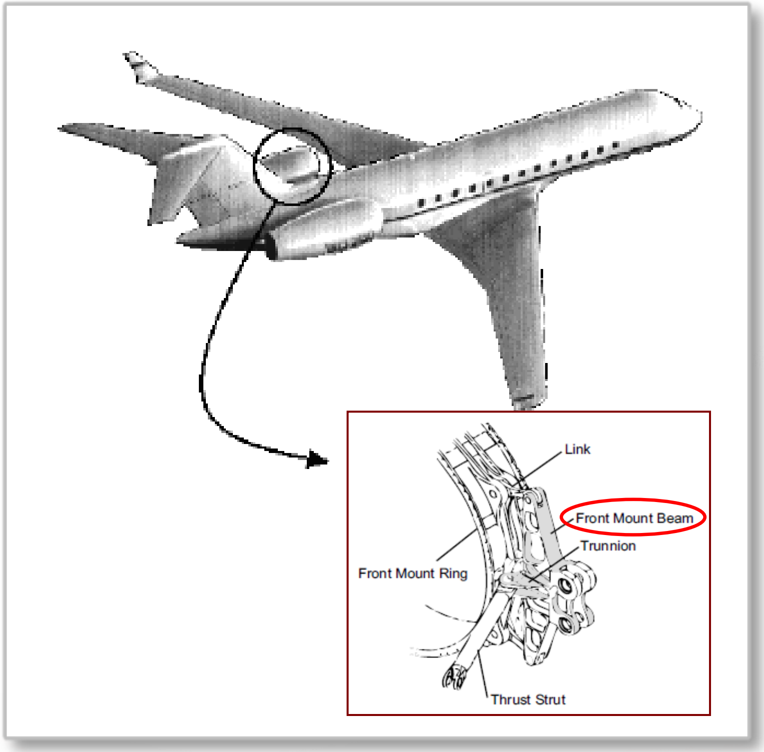
Topology Optimization Real Life Example

Wheel Topology Optimization – Proposed Solution



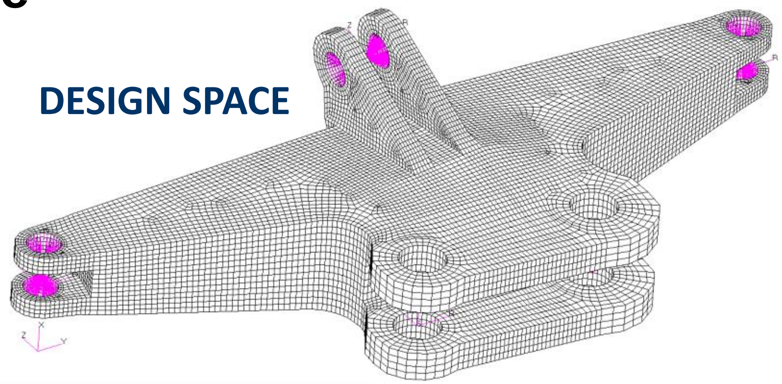
Topology Optimization Real Life Example
Airplane Engine Mount Topology Optimization

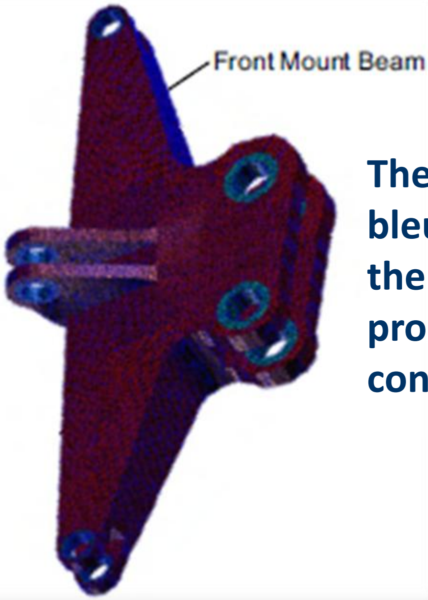
- **The main objective is to minimize the compliance of the front Mount Beam of an airplane engine**



14 load conditions relative to the several combinations of the loads acting in the connecting points and due to the most critical manouvres

DESIGN SPACE



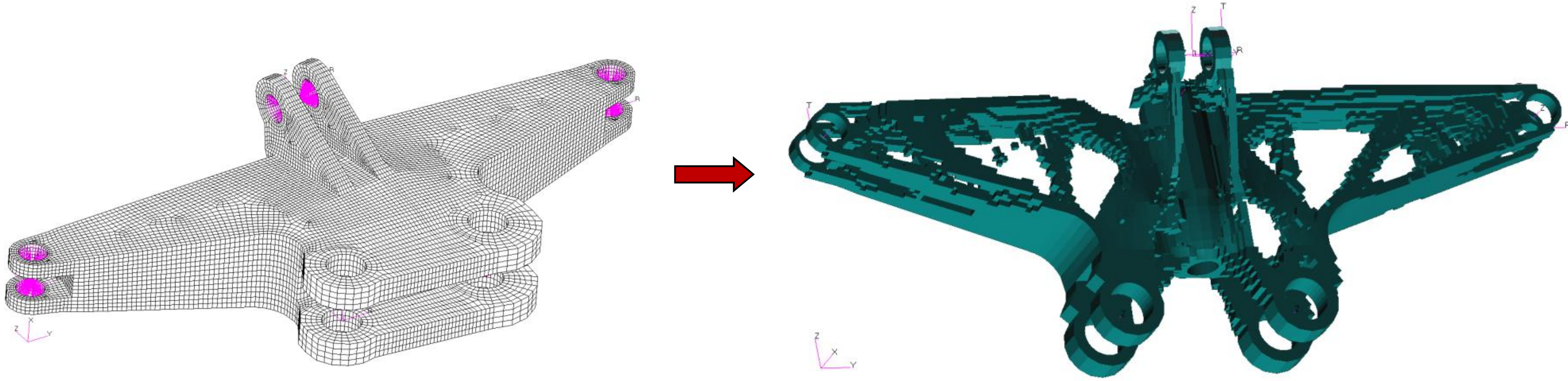


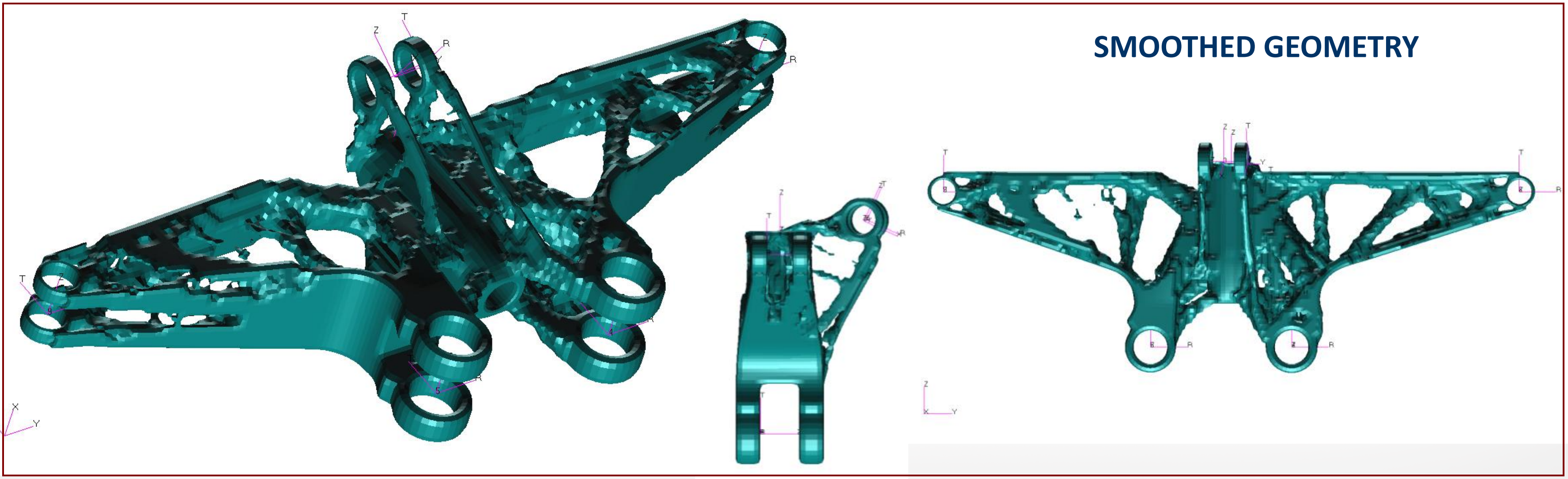
The regions represented in bleu must not be modified by the topological optimization process because they are the constraints locations

Topology Optimization Real Life Example

Airplane Engine Mount Topology Optimization – Proposed Solution

- **Case 1 – Without manufacturability constraints**

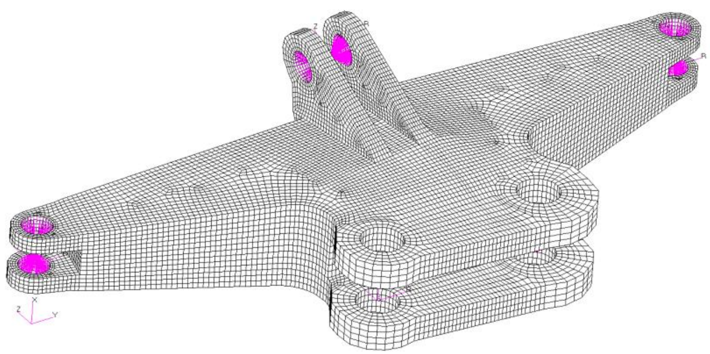


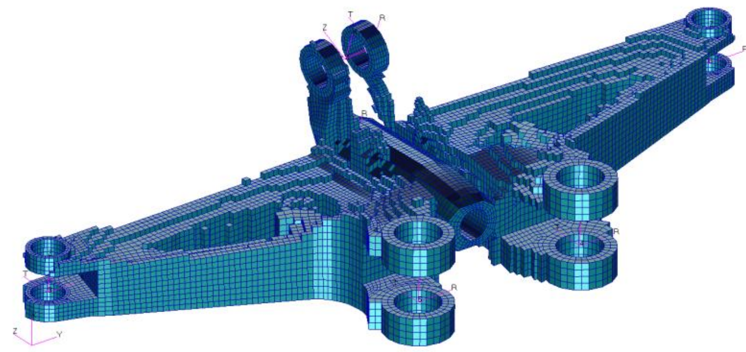


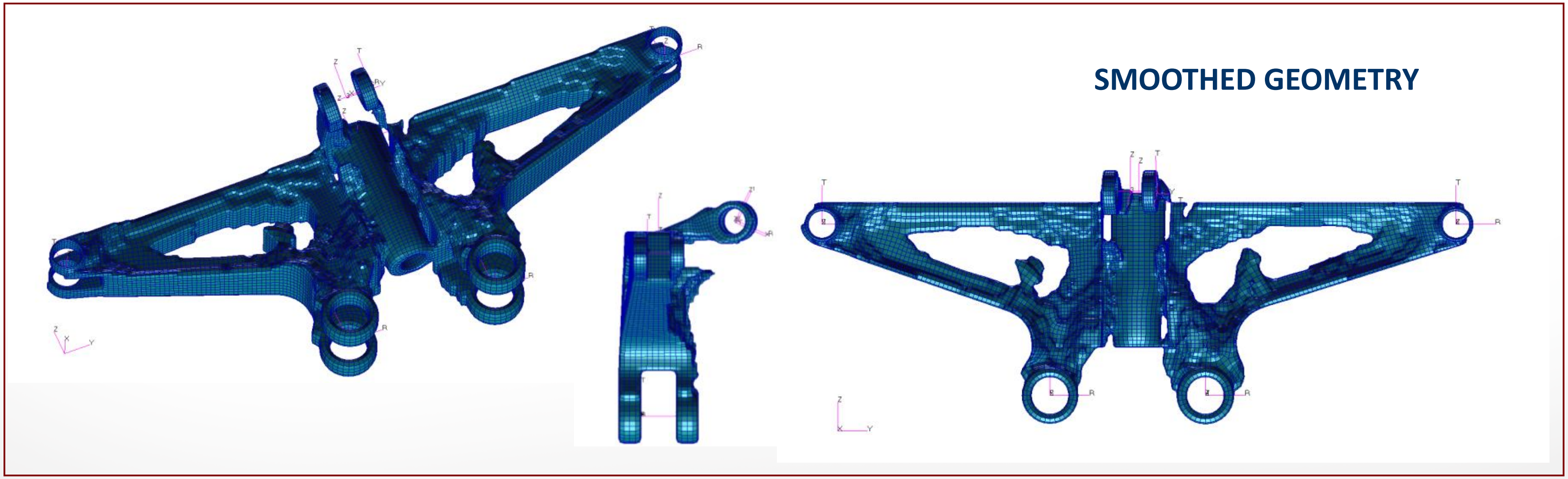
Topology Optimization Real Life Example

Airplane Engine Mount Topology Optimization – Proposed Solution

- **Case 1 – With manufacturability constraints (Casting)**

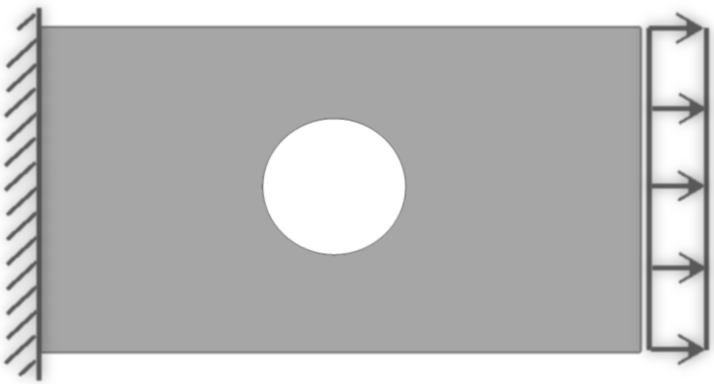






Shape Optimization

Shape Optimization
What is shape optimization?



- **It is the optimization of the boundary profile of a structural component typically used to reduce stresses or make the stress distribution uniform without increasing the weight of the structure.**
- **The boundary could be external as in a fillet shape design or internal as in a hole profile design**

- **PROBLEMS**

  Finite element model is not a CAD model

  – Parameters like radius or diameter are unknown in a finite element model
  – Only coordinates of the grid points defining the finite element model are known

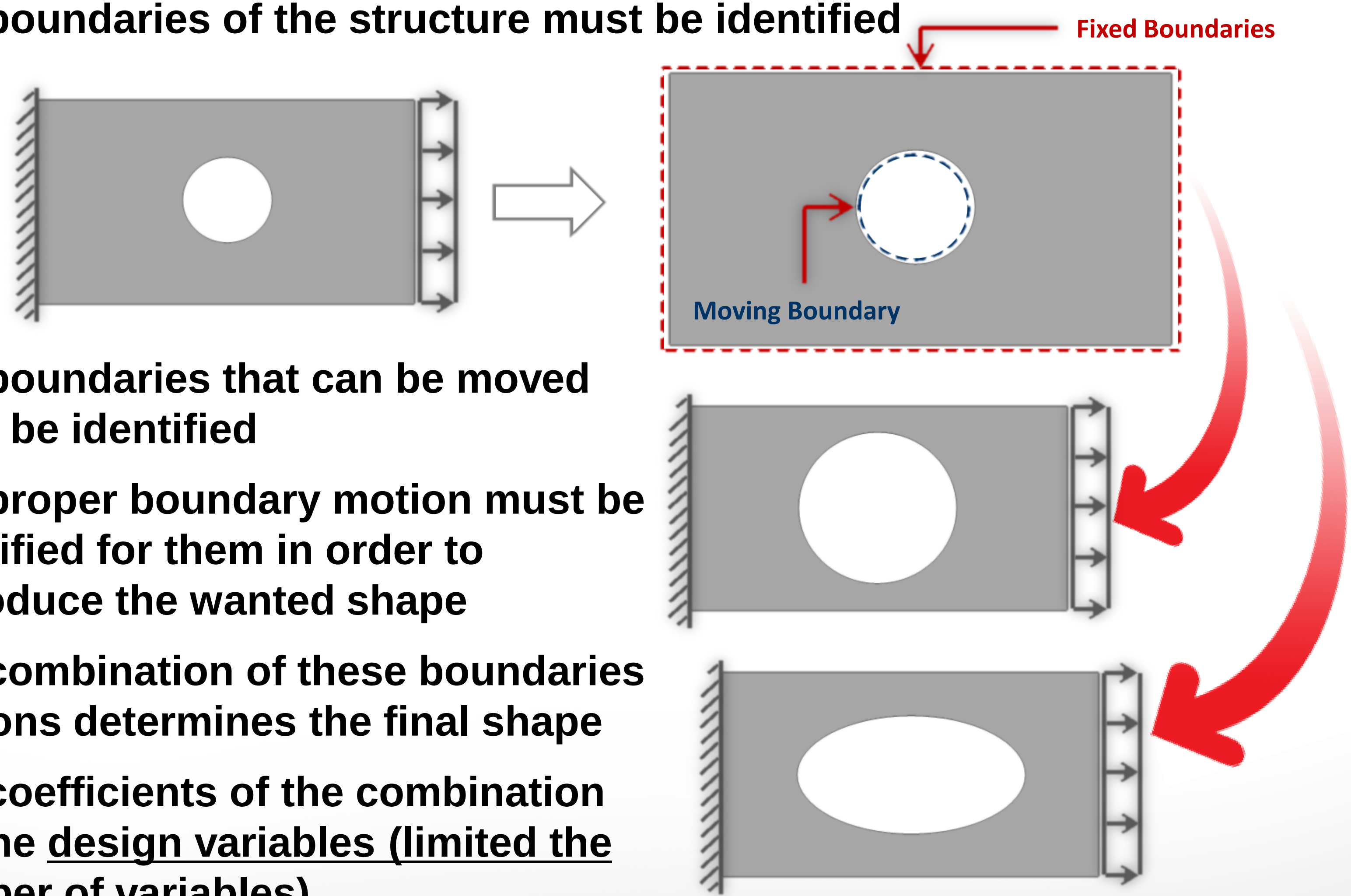
  The grid points coordinates are the real design variables

  – Too much design variables → 3 design variables for each grid point

- **A specific procedure must be considered to get over them**

Shape Optimization
Problem Definition

- **The boundaries of the structure must be identified**



- **The boundaries that can be moved must be identified**
- **The proper boundary motion must be identified for them in order to reproduce the wanted shape**
- **The combination of these boundaries motions determines the final shape**
- **The coefficients of the combination are the <u>design variables (limited the number of variables)</u>**

Shape Optimization
Shape Basic Vectors Concept and Basic Equation

- **The main challenges are**
  a. Defining shape with a few parameters or shape design variables
  b. Re-meshing the domain upon change of boundary shape.
- **Both these are addressed in MD NASTRAN using the concept of shape basis vectors.**
- **Basis vectors can be understood as allowable shape changes or directions of movement of grids.**
  – While basis vectors give the direction of grid movement, design variables are used for determining the magnitude of change.

- **MSC NASTRAN uses shape basis vectors to describe shape changes**

$$\underbrace{\{\Delta G\}}_{\text{Grid Point Changes}} = \underbrace{[T]}_{\text{Basis Vector}} \cdot \underbrace{\{\Delta x\}}_{\text{Design Variable Changes}}$$

where

$$\left.\begin{aligned}\{\Delta G\} &= \{G\}_{i+1} - \{G\}_i \\ \{\Delta x\} &= \{x\}_{i+1} - \{x\}_i\end{aligned}\right. \quad \left\{\begin{aligned} &i \quad = \text{current design} \\ &i+1 = \text{updated design}\end{aligned}\right.$$

A new shape is a linear combination of basis vectors.

The engineer determines HOW the structure can change ⇒ [T]

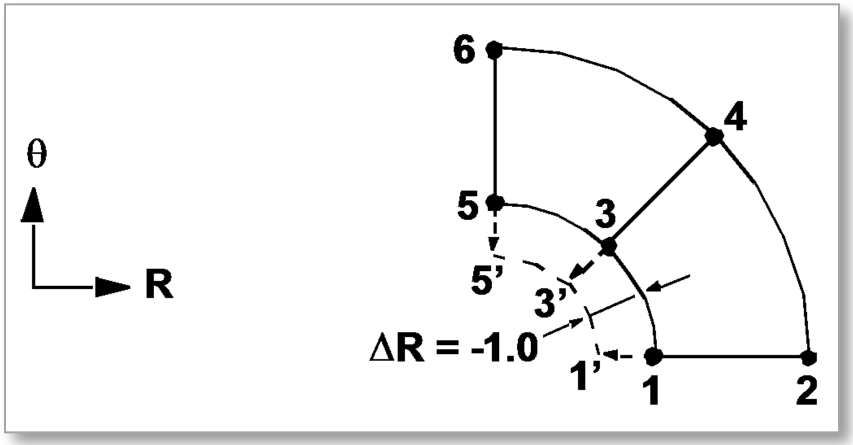
The optimizer determines HOW MUCH the structure can change ⇒[Δx]

Shape Optimization
Advantages of the method

- **Complex shapes can be defined using a linear combination of simple shapes (basis vectors)**
- **A large number of grids can be perturbed using a few design variables**
- **A number of methods are available to generate shape basis vectors which, though are independent of a pre-processor, can be very easily done through a pre-processor**
- **Since the basis vector serves the purpose of a displacement field for the entire grid set, there is no requirement of a separate re-meshing process**
- **If the basis vector is generated internally and is updated as the design changes, the problem of mesh distortion can be circumvented**

Shape Optimization
Basic vectors - Examples

- **Example 1 – Radius of Circular Plate**

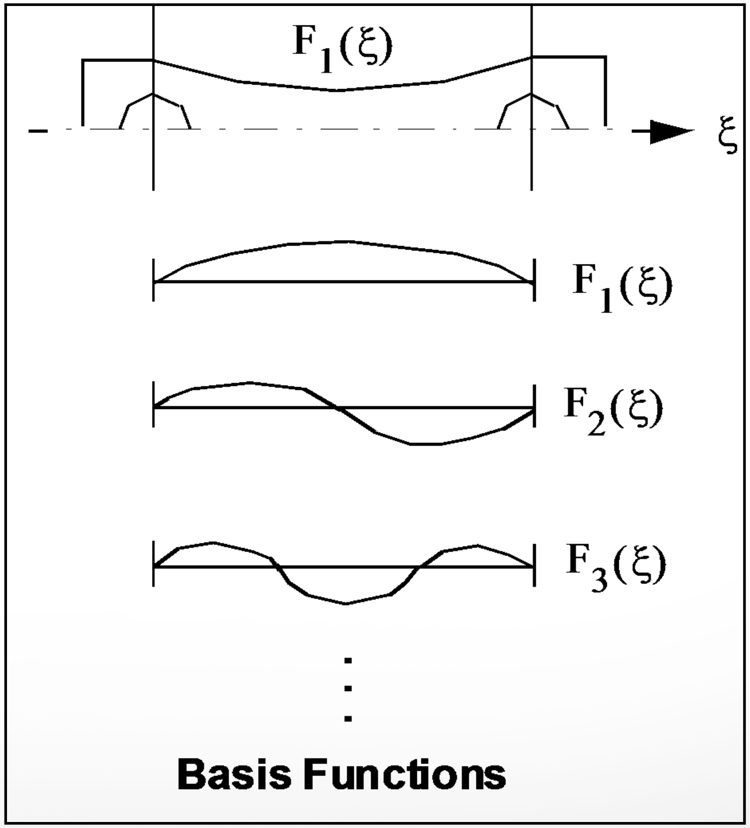


$$\{\Delta G\} = \{G'\} - \{G\} = \Delta x\{T\}$$

$$\{T\} = \left\{ \underbrace{-1.,0.,0.}_{GRID\ 1} \quad \underbrace{0.,0.,0}_{GRID\ 2} \quad \underbrace{-1.,0.,0.}_{GRID\ 3} \quad \underbrace{0.,0.,0.}_{GRID\ 4} \quad \underbrace{-1.,0.,0.}_{GRID\ 5} \quad \underbrace{0.,0.,0.}_{GRID\ 6} \right\}^T$$

- **Example 2**

$$F(\xi) = x_1 F_1(\xi) + x_2 F_2(\xi) + x_3 F_3(\xi)\ldots$$

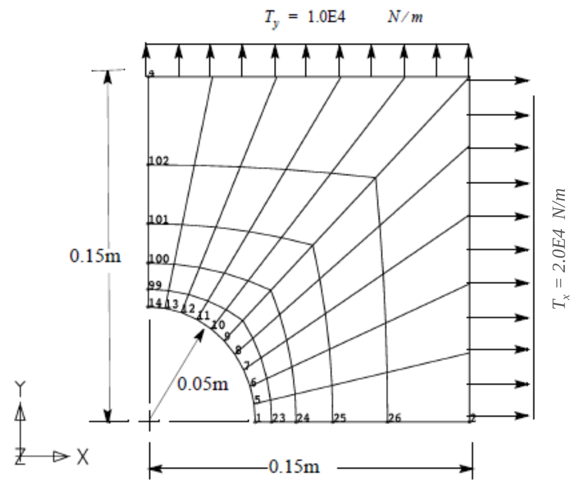


Shape Optimization
Basic vectors - Examples

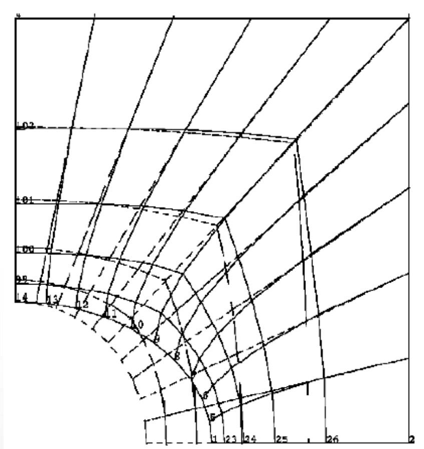
- **Example 3**
  - Consider a plate with a hole in the middle with unequal edge traction forces. A quarter model is pictured on the right:

    Material: Aluminum, 7075 –T6 Sheet

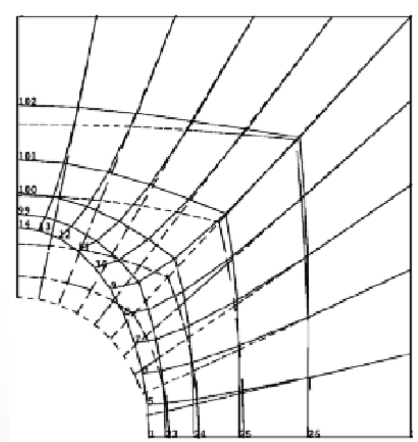
    E = 7.2 E10 N/m^2, ν = 0.33, ρ = 2.8 E3 kg/m^3



- **We expect the optimal hole shape to be an ellipse, so a candidate set of basis vectors might be:**



Basis Vector {T}$_1$



Basis Vector {T}$_2$

- **MSC/MD Nastran will allow us to define design variables that represent *x* and *y* axis intercepts as: $x_1 = a$, $x_2 = b$ so that,**

$$\{\Delta G\} = \left[\{T_1\} \{T_2\} \right] \begin{Bmatrix} \Delta x_1 \\ \Delta x_2 \end{Bmatrix} \begin{matrix} \leftarrow \Delta a \\ \leftarrow \Delta b \end{matrix}$$

x-direction variation

y-direction variation

Shape Optimization
Shape basic vectors in the design model

- **There are two primary tasks in design modeling for shape**
  - Defining the shape basis vectors
  - Defining the design variables and correlating these to the basis vectors
- **Defining shape basis vectors**
  - Each basis vector component describes the direction and magnitude of an individual grid component change, for a given design variable change
  - This is a large amount of data
  - MSC Nastran provides four methods of generating shape basis vectors
    - *Manual grid variation*
    - *Direct input of shapes*
    - *Geometric boundary shapes*
    - *Analytic boundary shapes*

Shape Optimization

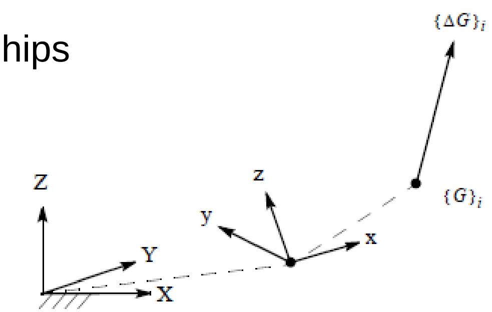
1 - Manual Grid Variation

- **This is the most general, as well as the most tedious, method of generating shape basis vectors**
  - It is recommended only for very simple cases, or in unique situations.
- **A DVGRID entry is used to define motion for a single grid:**

Change in design variable and not the absolute value of design variable relates to the change in grid position

(Design Variable to GRID) Relationships

$$\{\Delta G\}_i = \begin{Bmatrix}\Delta G_{ix}\\ \Delta G_{iy}\\ \Delta G_{iz}\end{Bmatrix} = COEFF\begin{Bmatrix}N1\\ N2\\ N3\end{Bmatrix}\cdot \Delta x_{DVID}$$



DVGRID	DVID	GID	CID	COEFF	N1	N2	N3		

- **Each DVGRID entry defines one to three basis vector components, as well as identifies the related design variable.**
  - The DVGRIDs referring to the same shape vector must refer to the same DESVAR (DVID)
- **Multiple references to the same grid-design variable result in vectorial addition of the basis vector component.**

Shape Optimization

1 - Manual Grid Variation – Patran shape utility

- **The method is symbiotic with some of the other methods**
  - the other methods are dependent on a few DVGRID entries and conversely,
  - the other methods greatly reduce the number of required DVGRID entries
- **Patran Shape Utility pre-processing feature makes the generation of a large number of DESVAR-DVGRID entries simple**
- **This utility is a way of creating Nastran shape optimization basis vectors using fields to define the shape change**
- **The user specifies which spatial vector fields to use and which grid points to move**

- **Basis vectors are written to a file dbname.bv.nn and an include statement is put into the current analysis direct text input to refer to this file**
- **Steps**
  - First create a spatial vector field for each basis vector
  - Then launch the Basis Vector Creation Utility Form

    Utilities → Analysis → Calculate Shape Basis Vectors
  - Select the fields that you want to use then select the nodes that you want to move

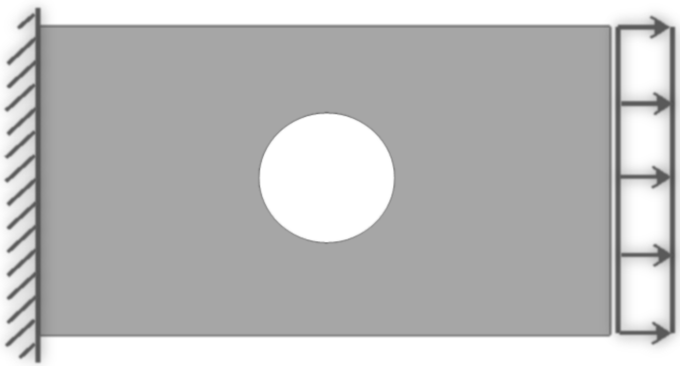
Shape Optimization
Auxiliary Models Concepts

- **Two principal issues using Maunual Grid Variation method**
  - Limiting the DVGRID entries to few grid points you can modify the elements distorting them too much
  - The grid points position should be modified with continuity in the regions between the moving and the fixed boundaries
- **The introduction of Auxiliary Models Concept helps in solving these issues reducing also the amount of data**
- **What is an auxiliary model?**
  - It is a structure whose deformations may be used to generate basis vectors.
  - Although the geometry is usually the same, boundary conditions, loads, and possibly material types may differ.
- **The following methods use auxiliary models as an aid to shape basis vector generation**
  - Direct input of shapes
  - Geometric boundary shapes
  - Analytic boundary shapes

Shape Optimization

Auxiliary Models Concepts (Cont.)

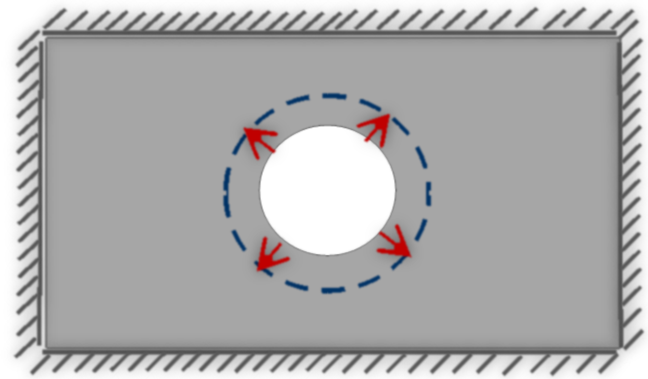
REAL MODEL



- **The edge on the left is fixed**
- **The loads are applied at the free end of the plate**

- **Loads applied to the hole boundary yield consistent deformations that can be used as basis vectors**
  - The grid points inside the structure are moved with continuity
- **The different methods differ for the way in which the shape vectors are built**

AUXILIARY MODEL



- **We are interested to modify the radius of the hole**
- **All the external edges (the edges of the rectangular region) are fixed**
- **The load consist of a unit radial displacement applied to all of the grid points located in the circular edge**

$\{U\}$ Vector from static analysis

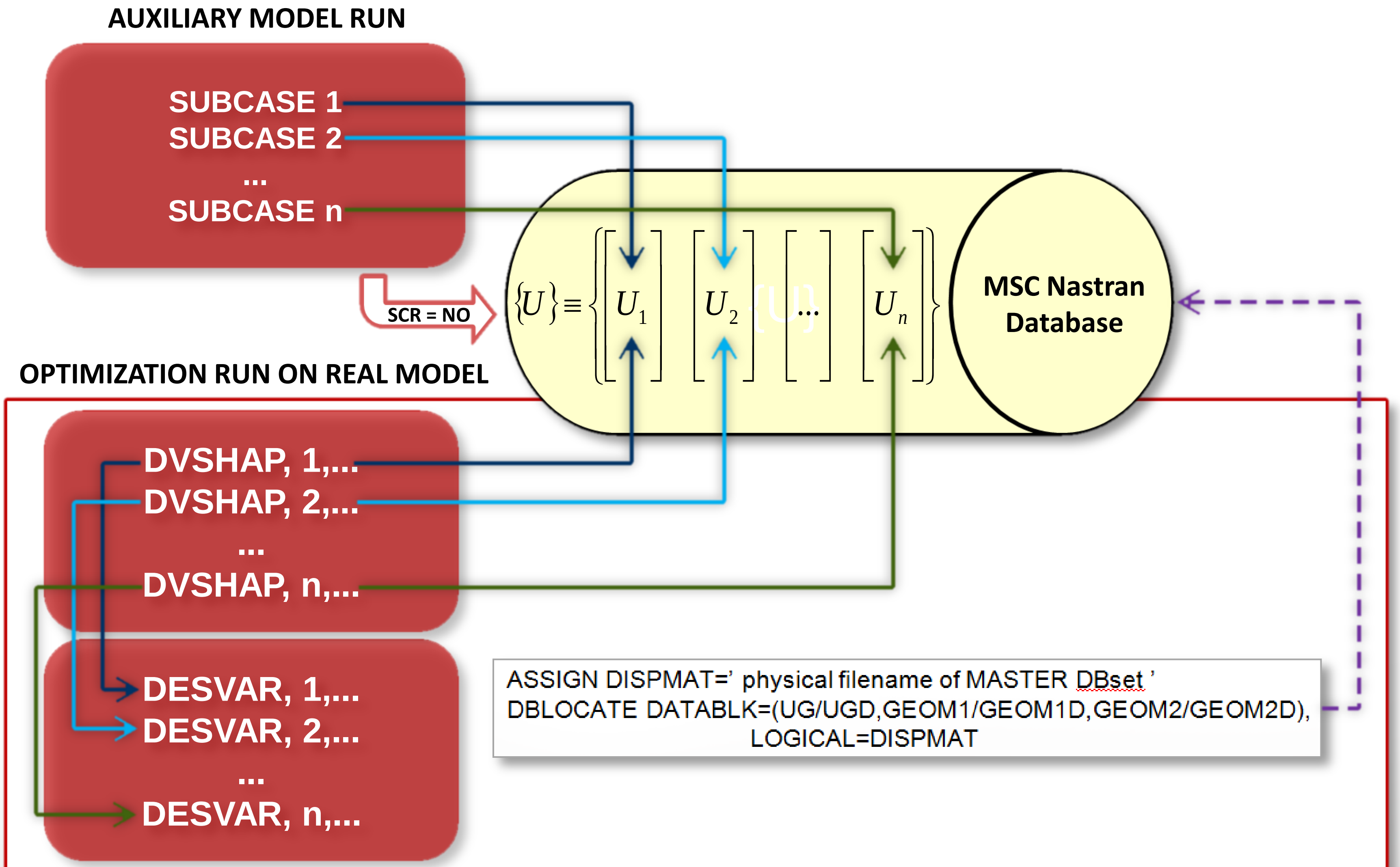
$$\{\Delta G\} = [T] \cdot \{\Delta x\}$$

Shape Optimization
2 - Direct Input of shapes

- **External auxiliary model is analyzed for different load cases**
  - It means that a separate run is executed
  - ‘scr = no’ must be defined to obtain a MSC Nastran database
- **Each column of the displacement vector corresponds to a load case and is used as a basis vector**
- **A number of DESVAR equal to the number of auxiliary model load case (to which we are interested) must be created**
- **DVSHAP Bulk Data Entry connects a DESVAR with a column number of the displacement vector**
- **The database in which the externally located displacement vectors are located is DBLOCATEd and used as basis vector**
- **Since basis vectors are externally generated, they are not updated in each design cycle and so mesh distortion may arise in case of large shape changes**

Shape Optimization

2 - Direct Input of Shapes (Graphical Representation)



Shape Optimization

2 - Direct Input of Shapes - DVSHAP entry

- **Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix**

Format:

1	2	3	4	5	6	7	8	9	10
DVSHAP	DVID	COL1	SF1	COL2	SF2	COL3	SF3		

Example:

DVSHAP	2	1	2.0	4	1.0				

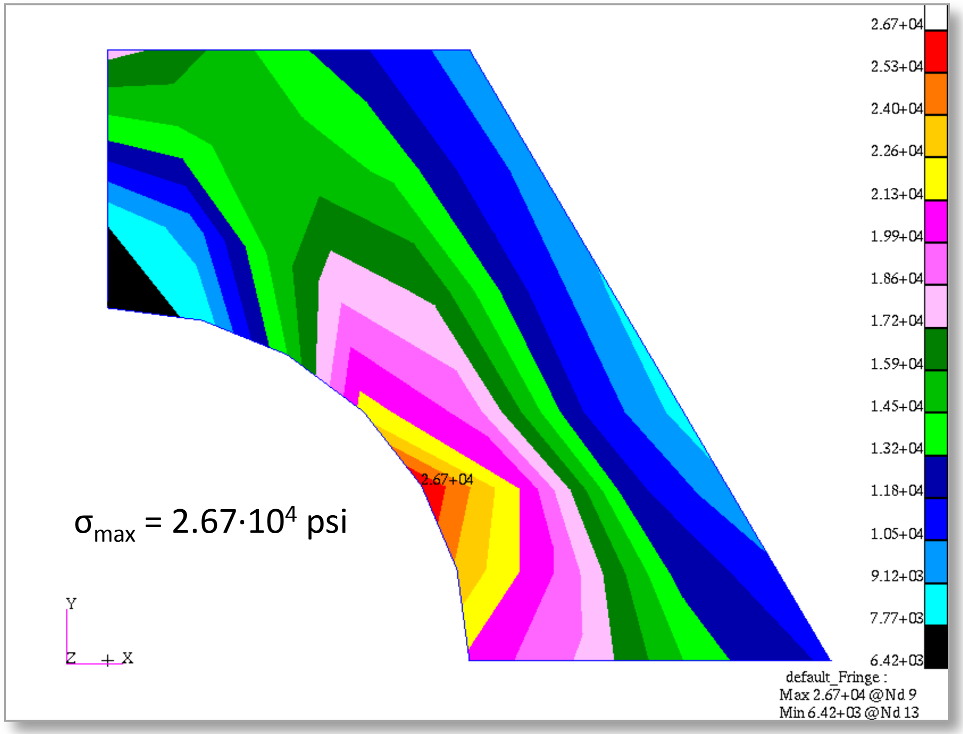
Field	Contents
DVID	Design variable identification number on the DESVAR entry. (Integer > 0)
COLi	Column number of the displacement matrix. See Remark 2. (1 ≤ Integer ≤ maximum column number in the displacement matrix.)
SFi	Scaling factor applied to the COLi-th column of the displacement matrix. (Real; Default = 1.0)

- **Remarks:**
  1. DVID must be defined on a DESVAR entry.
  2. COLi must be a valid column number in the displacement matrix.
  3. Multiple references to the same DVID and/or COL_i will result in a linear combination of displacement vectors. In the example above, the shape basis vector is a linear combination of the fourth column and twice the second column.
  4. The displacement matrix must have been created by MSC/MD Nastran and be available on a database, which is attached via the DBLOCATE FMS statement shown below:

```
ASSIGN DISPMAT=' physical filename of MASTER DBset '
DBLOCATE DATABLK=(UG/UGD,GEOM1/GEOM1D,GEOM2/GEOM2D),
                                                    LOGICAL=DISPMAT
```

Shape Optimization
2 - Direct Input of Shapes: Simple Example

- **TPL problems d200csx.dat, d200cs.dat**
- **The figure below is an initial design with von Mises stress resultants for a two-dimensional, symmetric road support structure (culvert).**
- **From the stress distribution on the circular boundary, we can observe that the culvert interior profile is not optimal.**

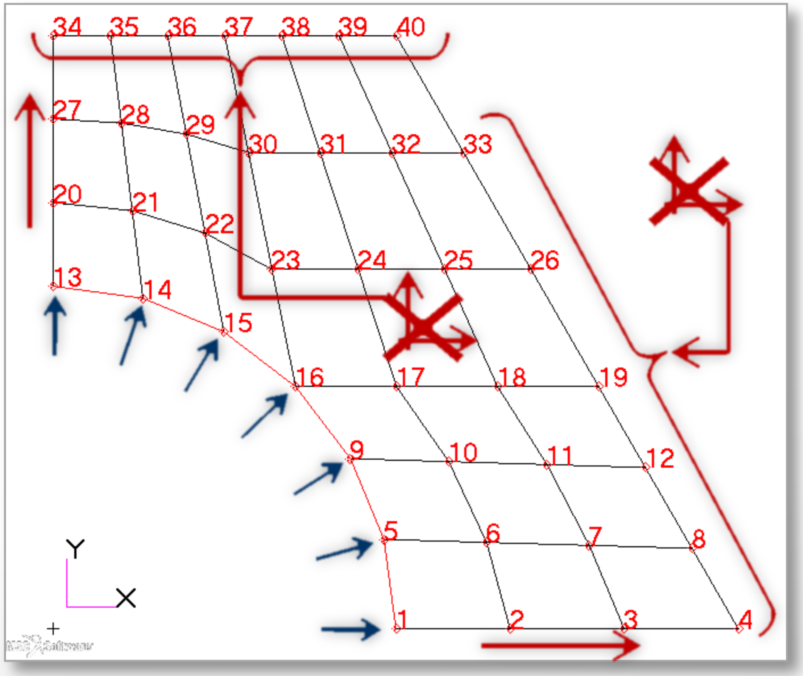


- **Shape Optimal Solution Approach**
  - Create auxiliary model and solve for desired shape changes
  - Gather displacement solutions to use as basis vectors
  - Define a design task

Shape Optimization

2 - Direct Input of Shapes: Simple Example (Auxiliary Model)

- **We require that the outer culvert profile remain unchanged.**
  - Outside edges of the culvert are fixed in the auxiliary model to satisfy straight edge requirements.
  - Grid points 1, 2, 3 on the bottom are allowed to move in the x-direction
  - Grid points 13, 20, 27 on the symmetry line can move along the y-direction.
- **Interior shape variations can be achieved by loading each interior grid individually**
  - Grids 5, 9, 14, 15, and 16 on the hole boundary are allowed to move as well.
  - Along this hole boundary, six CBAR elements have been added to help smooth the applied loading effects.
    - It is important to allow z-rotations along this boundary.
- **To generate a set of displacements to be used as basis vectors, we can statically load each grid separately in a direction normal to the boundary**
  - 7 separate load conditions as the number of the grid points along the hole edge.



Shape Optimization

2 - Direct Input of Shapes: Example (Run Auxiliary Analysis)

Submit the auxiliary job with keyword scr=no to store displacement vectors in the Nastran database.

```
ID,AUX1,VT100 $FEB 11,2008
TIME 10
SOL 101
CEND
TITLE=Culvert Example Using External Auxil
SUBTITLE=The External Auxiliary Model
SPC=25
$
$ seven load cases
$
SUBCASE 1
 LOAD=100
 DISP=ALL
SUBCASE 2
 LOAD=101
 DISP=ALL
SUBCASE 3
 LOAD=102
 DISP=ALL
SUBCASE 4
 LOAD=103
 DISP=ALL
SUBCASE 5
 LOAD=104
 DISP=ALL
SUBCASE 6
 LOAD=105
 DISP=ALL
SUBCASE 7
 LOAD=106
 DISP=ALL
BEGIN BULK
PARAM,POST,0
param,newseq,-1
$
$ The same GRID and CQUAD4 entries as the
$ primary structure
$
```

```
$
GRID,  1,, 3.00000, 0.00000,.00
GRID,  2,, 4.00000, 0.00000,.00
GRID,  3,, 5.00000, 0.00000,.00
  .    .     .       .      .
 (see optimization input file)
.    .      .       .      .
GRID, 38,, 2.00000, 5.19600,.00
GRID, 39,, 2.50000, 5.19600,.00
GRID, 40,, 3.00000, 5.19600,.00
CQUAD4,     1,101,     1,     2,     6,     5
CQUAD4,     2,101,     2,     3,     7,     6
CQUAD4,     3,101,     3,     4,     8,     7
  .         .  .       .      .      .      .
 (see optimization input file)
  .         .  .       .      .      .      .
CQUAD4,    25,101,    30,    31,    38,    37
CQUAD4,    26,101,    31,    32,    39,    38
CQUAD4,    27,101,    32,    33,    40,    39
PSHELL,101,102,.44
MAT1,102,2.+7,,.3
$
$ Additional CBAR elements maintain smoothness
$ of the circular boundary
$
CBAR,31,1,13,14,,1.0
CBAR,32,1,14,15,,1.0
CBAR,33,1,15,16,,1.0
CBAR,34,1,16, 9,,1.0
CBAR,35,1, 9, 5,,1.0
CBAR,36,1,5 , 1,,1.0
PBAR     1        102      20.0     1.0      1.0
$
 .         .  .       .      .      .      .
```

```
$ Seven load cases
$
FORCE,100,13,0,1.e5,0.,1.,0.
FORCE,101,14,0,1.e5,0.259,.9659
FORCE,102,15,0,1.e5,0.5,0.866,0.0
FORCE,103,16,0,1.e5,1.,1.,0.
FORCE,104,9,0,1.e5,0.866,0.5,0.0
FORCE,105,5,0,1.e5,0.9659,0.259
FORCE,106,1,0,1.e5,1.,0.,0.
$
$ Boundary conditions satisfy functional
$ and $ manufacturing requirements
$
SPC1,25,345,1,THRU,40
SPC1,25,6,2,THRU,4
SPC1,25,6,6,THRU,8
SPC1,25,6,10,THRU,12
SPC1,25,6,17,THRU,19
SPC1,25,6,20,THRU,26
SPC1,25,6,27,THRU,33
SPC1,25,6,34,THRU,40
SPC1,25,12,33,THRU,40
SPC1,25,12,4,8,12,19,26
SPC1,25,1,13,20,27
SPC1,25,2,1,2,3
ENDDATA
```

Shape Optimization

2 - Direct Input of Shapes: Example (Optimization Run)

```
$
$ FMS section for retrieving the auxiliary displacement
matrix
$
assign f1_aux='culvert1.MASTER'
dblocate datablk=(ug/ugd,geom1/geom1d,geom2/geom2d) ,
 logical=f1_aux
SOL    200  $
TIME   100
CEND
TITLE=CULVERT EXAMPLE USING EXTERNAL AUXILIARY STRUCTURE
SUBTITLE=THE PRIMARY STRUCTURE
ANALYSIS = STATICS
SPC=25
 LOAD=1
 DISP=ALL
 STRESS=all
 DESSUB = 10
 desobj = 5
BEGIN BULK
PARAM,POST,0
$ PARAM,optexit,4
PARAM,NEWSEQ,-1
GRID,  1,, 3.00000, 0.00000,.00
GRID,  2,, 4.00000, 0.00000,.00
GRID,  3,, 5.00000, 0.00000,.00
GRID,  4,, 6.00000, 0.00000,.00
GRID,  5,, 2.89464, 0.78478,.00
```

- **Shape design task**
  - Objective: minimizing volume of the culvert
  - Constraints: Von Mises Stress <=31000
  - Shape design variables:
    - DBLOCATE the auxiliary displacement vectors
    - Define them as basis vectors using DESVAR and DVSHAP Bulk Data entries.

Shape Optimization

2 - Direct Input of Shapes: Example (Optimization Run)

```
GRID,  6,, 3.79369, 0.75885,.00
GRID,  7,, 4.69274, 0.73293,.00
GRID,  8,, 5.59178, 0.70700,.00
GRID,  9,, 2.60164, 1.49178,.00
GRID, 10,, 3.46229, 1.46585,.00
GRID, 11,, 4.32293, 1.43993,.00
GRID, 12,, 5.18357, 1.41400,.00
GRID, 13,, 0.00000, 3.00000,.00
GRID, 14,, 0.78478, 2.89464,.00
GRID, 15,, 1.49178, 2.60164,.00
GRID, 16,, 2.12100, 2.12100,.00
GRID, 17,, 3.00578, 2.12100,.00
GRID, 18,, 3.89057, 2.12100,.00
GRID, 19,, 4.77535, 2.12100,.00
GRID, 20,, 0.00000, 3.73200,.00
GRID, 21,, 0.68985, 3.66176,.00
GRID, 22,, 1.32785, 3.46643,.00
GRID, 23,, 1.91400, 3.14600,.00
GRID, 24,, 2.67052, 3.14600,.00
GRID, 25,, 3.42704, 3.14600,.00
GRID, 26,, 4.18357, 3.14600,.00
GRID, 27,, 0.00000, 4.46400,.00
GRID, 28,, 0.59493, 4.42888,.00
GRID, 29,, 1.16393, 4.33122,.00
GRID, 30,, 1.70700, 4.17100,.00
GRID, 31,, 2.33526, 4.17100,.00
GRID, 32,, 2.96352, 4.17100,.00
GRID, 33,, 3.59178, 4.17100,.00
GRID, 34,, 0.00000, 5.19600,.00
GRID, 35,, 0.50000, 5.19600,.00
GRID, 36,, 1.00000, 5.19600,.00
GRID, 37,, 1.50000, 5.19600,.00
GRID, 38,, 2.00000, 5.19600,.00
GRID, 39,, 2.50000, 5.19600,.00
GRID, 40,, 3.00000, 5.19600,.00
```

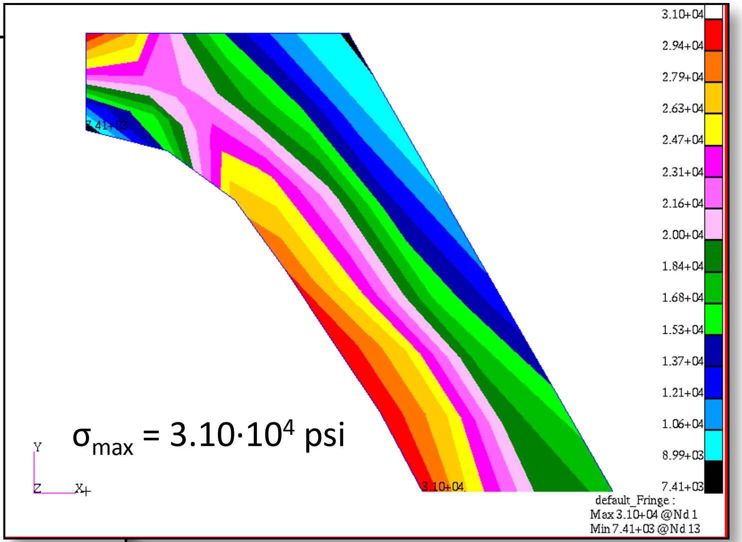
```
CQUAD4,    1,101,    1,    2,    6,    5
CQUAD4,    2,101,    2,    3,    7,    6
CQUAD4,    3,101,    3,    4,    8,    7
CQUAD4,    4,101,    5,    6,   10,    9
CQUAD4,    5,101,    6,    7,   11,   10
CQUAD4,    6,101,    7,    8,   12,   11
CQUAD4,    7,101,    9,   10,   17,   16
CQUAD4,    8,101,   10,   11,   18,   17
CQUAD4,    9,101,   11,   12,   19,   18
CQUAD4,   10,101,   13,   14,   21,   20
CQUAD4,   11,101,   14,   15,   22,   21
CQUAD4,   12,101,   15,   16,   23,   22
CQUAD4,   13,101,   20,   21,   28,   27
CQUAD4,   14,101,   21,   22,   29,   28
CQUAD4,   15,101,   22,   23,   30,   29
CQUAD4,   16,101,   27,   28,   35,   34
CQUAD4,   17,101,   28,   29,   36,   35
CQUAD4,   18,101,   29,   30,   37,   36
CQUAD4,   19,101,   16,   17,   24,   23
CQUAD4,   20,101,   17,   18,   25,   24
CQUAD4,   21,101,   18,   19,   26,   25
CQUAD4,   22,101,   23,   24,   31,   30
CQUAD4,   23,101,   24,   25,   32,   31
CQUAD4,   24,101,   25,   26,   33,   32
CQUAD4,   25,101,   30,   31,   38,   37
CQUAD4,   26,101,   31,   32,   39,   38
CQUAD4,   27,101,   32,   33,   40,   39
FORCE   1       34      0       1250.           -1.
FORCE   1       35      0       2500.           -1.
FORCE   1       36      0       2500.           -1.
FORCE   1       37      0       2500.           -1.
FORCE   1       38      0       2500.           -1.
FORCE   1       39      0       2500.00         -1.
FORCE   1       40      0       1250.           -1.
```

Shape Optimization

2 - Direct Input of Shapes: Example (Optimization Run /Results)

```
PSHELL,101,102,.44
MAT1,102,2.+7,,.3,0.731-3
SPC1,25,3456,1,THRU,40
SPC1,25,12,1,THRU,4
SPC1,25,1,13,20,27,34
$
$ design model
$
desvar  1       b1      3.      -1.e6       1.e6     .2
desvar  2       b1      3.      -1.e6       1.e6     .2
desvar  3       b3      3.      -1.e6       1.e6     .2
desvar  4       b4      3.      -1.e6       1.e6     .2
desvar  5       b5      3.      -1.e6       1.e6     .2
desvar  6       b6      3.      -1.e6       1.e6     .2
desvar  7       b7      3.      -1.e6       1.e6     .2
$
$ A DVSHAP entry defines a shape basis vector by associating one design
$ variable to a dblocated displacement vector.
$
dvshap  1       1       66.773
dvshap  2       2       117.35
dvshap  3       3       216.33
dvshap  4       4       443.55
dvshap  5       5       220.89
dvshap  6       6       115.69
dvshap  7       7       65.669
dresp1  5       volume  volume
dresp1  2       von-mis stress  pshell          9               101
DCONSTR 10      2       -3.100e43.100e4
doptprm DESMAX  25      APRCOD  1
param,nasprt,1
ENDDATA
```

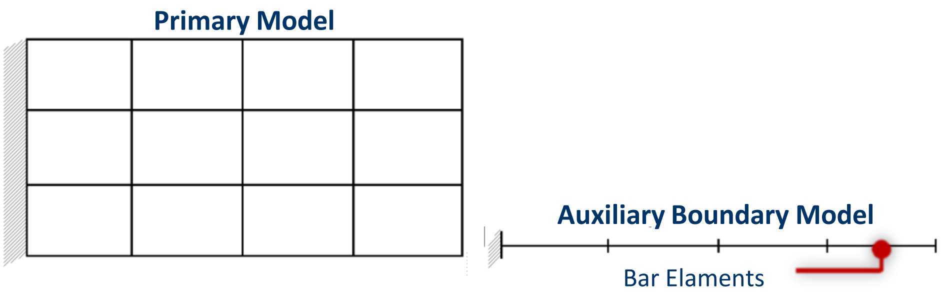
These scaling factors are chosen to normalize the basis vectors with the resulting maximum component of each basis vectors being unit.

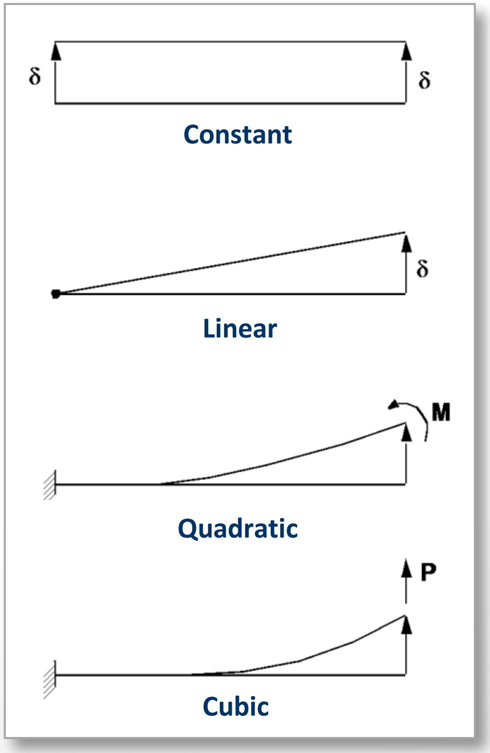


- **Results**
  - Final Volume is reduced by more than 20%. Maximum stress is increased by 16% and satisfies imposed limit

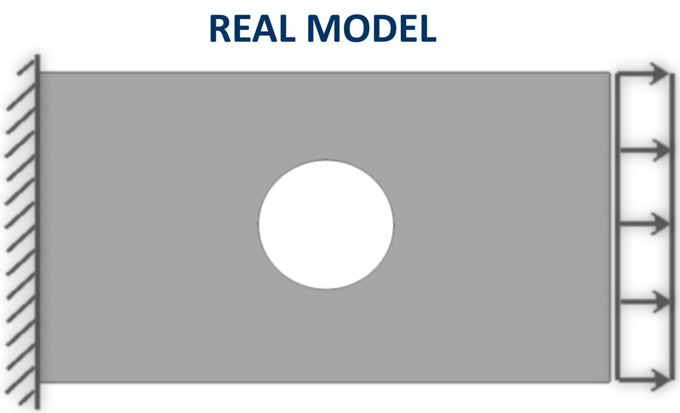
Shape Optimization
Auxiliary Boundary Models Concept

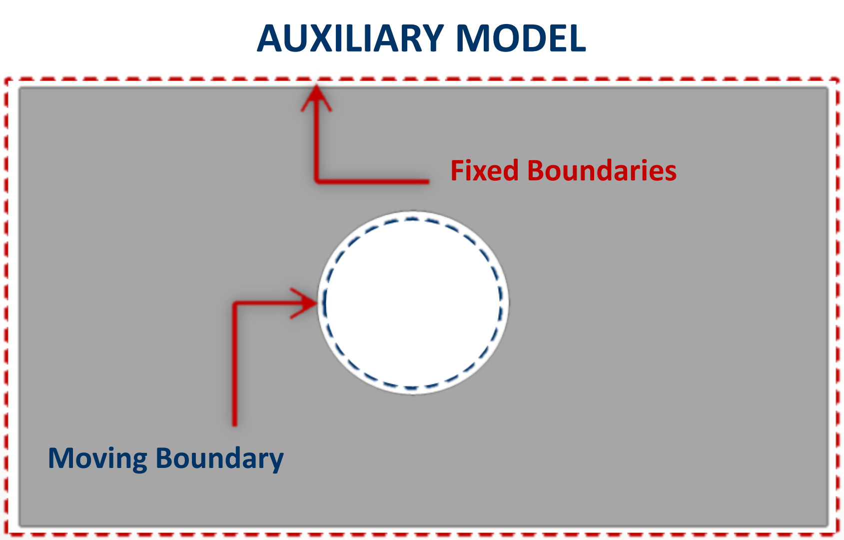
- **We have seen how auxiliary models can be used to generate shape variations over the domain of the structure.**
- **Auxiliary boundary models can likewise be used to generate shape variations over the boundaries of structures.**
  - These can be used to smooth out the applied shape-changing loads.
  - The <u>geometric boundary shapes</u> and <u>analytic boundary shapes</u> methods, both interpolate boundary variations to the structure’s interior.
  - The result is a set of shape basis vectors.
- **The two methods differ in the auxiliary boundary models definition**





Shape Optimization
3 – Geometry Boundary Shapes





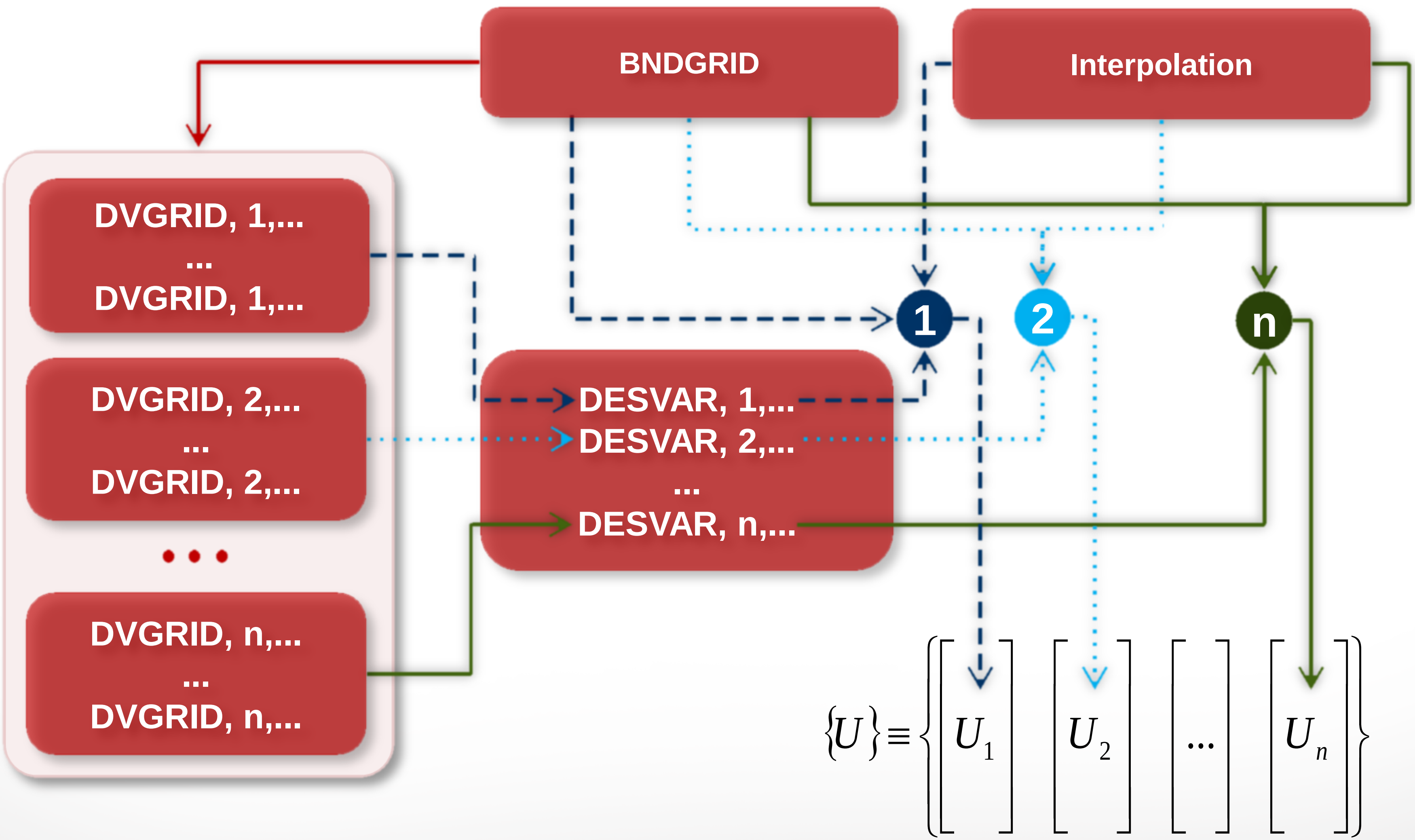
- **Boundary Definition**
  - Both moving and fixed boundaries
- **Identification of the moving boundaries**
  - The defined displacements are applied to these boundaries
  - The degrees of freedom defined as boundaries but not declared as moving are considered as fixed in the auxiliary model
- **Note that all or a subset of the degrees of freedom of a grid can be defined as part of the boundary**
- **The motion of all the missing degrees of freedom are moved by interpolation**

Shape Optimization
3 – Geometry Boundary Shapes

- **The boundary grids are identified using BNDGRID entries, that identify two categories of grid components:**
  - Grid components that change according to data supplied on the DVGRID entries
  - Grid components that remain fixed.
- **The DVGRID entry supplies the grid motion on the boundary**
- **The boundary conditions defined by the DVGRID and BNDGRID entries are only used to generate shape basis vectors. They are not used in connection with the primary model analysis.**
  - The movement of the interior grid points are automatically interpolated from the boundary
- **Since the basis vector is generated internally, they can be recomputed after each design cycle and thus there is little chance of mesh distortion with large shape changes**

Shape Optimization

3 – Geometry Boundary Shape (Graphical Representation)



Shape Optimization

3 – Geometry Boundary Shape: BNDGRID Entry

- **Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200)**

Format:

1	2	3	4	5	6	7	8	9	10
BNDGRID	C	GP1	GP2	GP3	GP4	GP5	GP6	GP7	
	GP8	-etc.-							

Example:

BNDGRID	123	41	42	43	44	45	46	47	
	49								

Alternate Format and Example:

BNDGRID	C	GP1	"THRU"	GP2					
BNDGRID	123	41	THRU	49					

Field	Contents
C	Component number (any unique combination of integers 1 through 6 with no embedded blanks). See Remark 1.
GPi	Shape boundary grid point identification number. (0 < Integer < 1000000; For THRU option, GP1 < GP2)

Remarks:

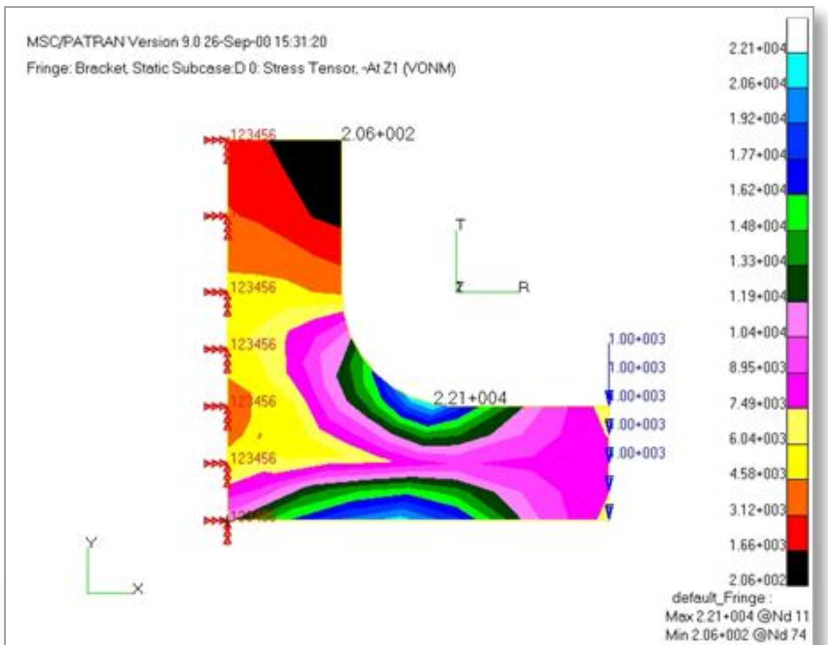
1. C specifies the components for the listed grid points for which boundary motion is prescribed.
2. Multiple BNDGRID entries may be used to specify the shape boundary grid point identification numbers.
3. Both, fixed and free shape boundary grid point identification numbers are listed on this entry.
4. The degrees of freedom specified on BNDGRID entries must be sufficient to statically constrain the model.
5. Degrees of freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See the *MSC Nastran Quick Reference Guide*, Appendix B for a list of these entries.

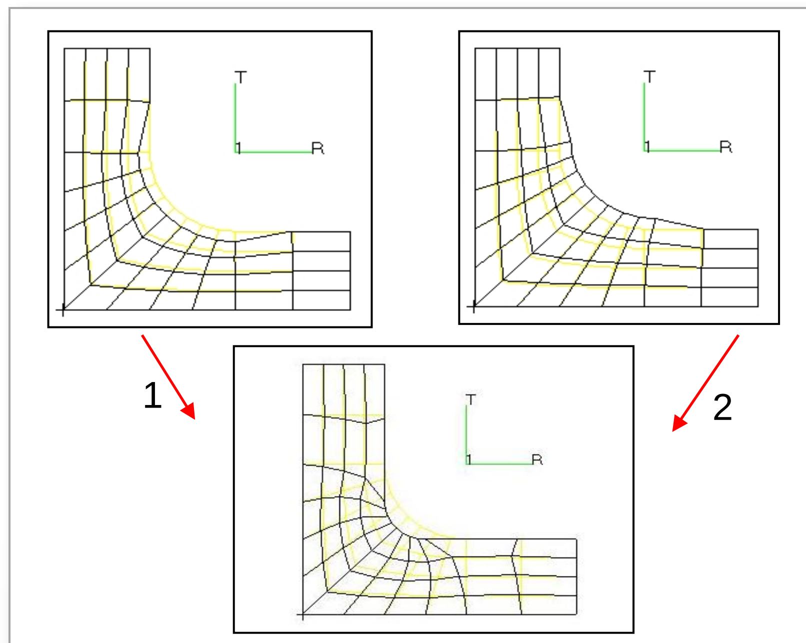
Shape Optimization

3 – Geometry Boundary Shape: Example

- **Design model description**
  - Design Variable - Radius of the Fillet
  - Objective - Weight
  - Constraints - Von Mises Stress < 1.75E4

- **The challenging job is to generate a basis vector that changes radius of the fillet and maintains the fillet tangency.**



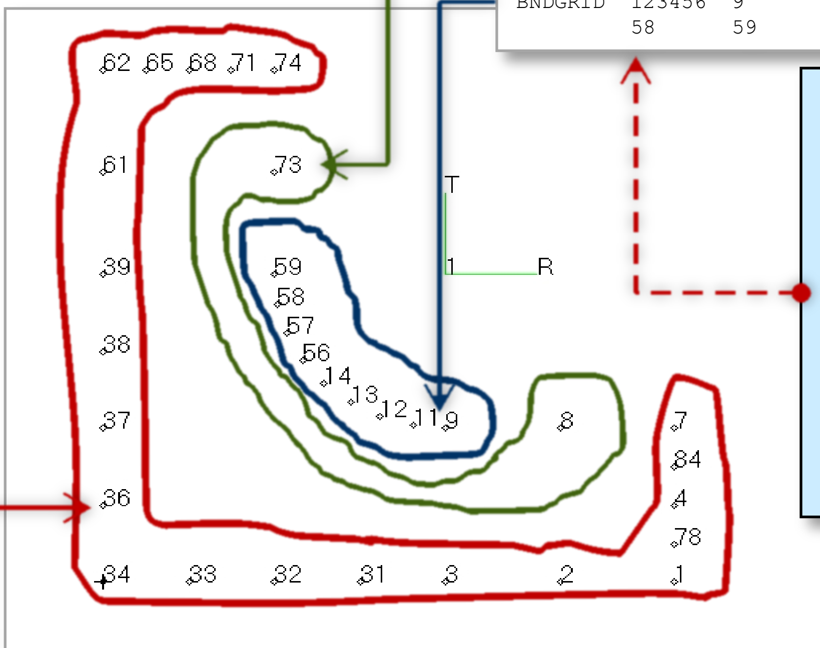


Shape Optimization

3 – Geometry Boundary Shape: Example (Identify boundary grids)

```
Identify grids whose motions are to be interpolated.
BNDGRID  23456  8
BNDGRID  13456  73
```

```
Identify grids on the circular boundary
BNDGRID  123456  9       11      12      13      14      56      57
                 58      59
```




```
Basis vector defining radial changes.
$
$  Define radial motion for the grids on the arc with a cylindrical
system
$
$DVGRID  DVID    GRIDID  CPID    COEFF   N1      N2      N3
$
DVGRID   1       59      1       1.      1.
DVGRID   1       58      1       1.      1.
DVGRID   1       57      1       1.      1.
DVGRID   1       56      1       1.      1.
DVGRID   1       14      1       1.      1.
DVGRID   1       13      1       1.      1.
DVGRID   1       12      1       1.      1.
DVGRID   1       11      1       1.      1.
DVGRID   1       9       1       1.      1.
```

```
Basis vector maintaining tangency.
$
$ Motion for maintaining tangency between the fillet and straight lines
$
DVGRID   2       59      1.      1.      1.
DVGRID   2       58      1.      1.      1.
DVGRID   2       57      1.      1.      1.
DVGRID   2       56      1.      1.      1.
DVGRID   2       14      1.      1.      1.
DVGRID   2       13      1.      1.      1.
DVGRID   2       12      1.      1.      1.
DVGRID   2       11      1.      1.      1.
DVGRID   2       9       1.      1.      1.
$
```

```
Identify grids that defines the coordinate system.
BNDGRID  123456  1111    1112    1113
```

```
$ Identify fixed boundary grid points using BNDGRID
BNDGRID  123456  1       2       3       4       7       31      32
                 33      34      36      37      38      39      61      62
                 65      68      71      74      78      84
```

Shape Optimization

3 – Geometry Boundary Shape: Example (Input File)

```
$
======================================================================
$     The purpose of this deck is to generate shape basis vectors
$     using the Geometric Boundary Shape Approach
======================================================================
$
SOL    200  $ Shape Optimization of a Bracket
CEND
TITLE = Bracket with Varying radius and center
SUBTITLE = Special Design Modeling Technique
ANALYSIS = STATICS
DISP = ALL
STRESS = ALL
SPC = 1
LOAD = 2
DESOBJ = 100
DESSUB = 1
BEGIN BULK
$
DESVAR  1       radius  2.0     1.0     3.0
DESVAR  2       radius  2.0     1.0     3.0
$
$ Identify fixed boundary grid points
$
BNDGRID 123456  1       2       3       4       7       31      32
        33      34      36      37      38      39      61      62
        65      68      71      74      78      84
$
$ Identify grid points on the circular boundary
$
BNDGRID 123456  9       11      12      13      14      56      57
                58      59
$
$ Identify grid points whose motions to be interpolated
BNDGRID 23456   8
BNDGRID 13456   73
$
$ Define various motions using DVGRID entries
$
$ First, create a cylindrical coordinate system to facilitate
$ specification of DVGRID entries for radial motion.
$ The center of the coordinate system is at (3.0,3.0,0.0)
$ These entries have been provided.
$
```

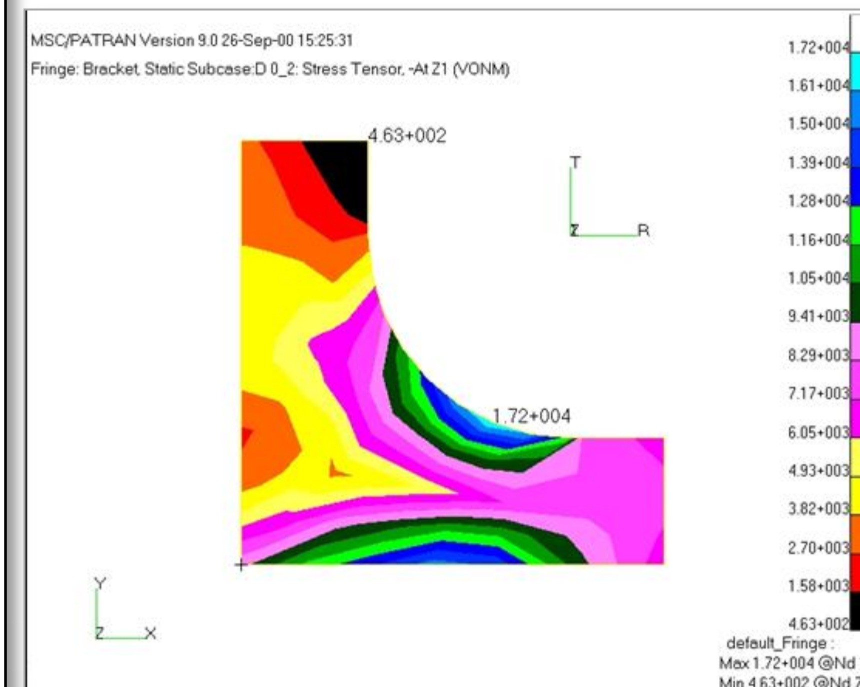
```
GRID    1112            3.0     3.0     1.0
GRID    1113            4.0     3.0     0.0
CORD1C  1       1111            1112    1113
GRID    1111            3.0     3.0     0.0
$
$ Define radial motion for the grids on the arc with a
$ cylindrical system
$
$DVGRID DVID    GRIDID  CPID    COEFF   N1      N2      N3
DVGRID    1     59      1       1.      1.
DVGRID    1     58      1       1.      1.
DVGRID    1     57      1       1.      1.
DVGRID    1     56      1       1.      1.
DVGRID    1     14      1       1.      1.
DVGRID    1     13      1       1.      1.
DVGRID    1     12      1       1.      1.
DVGRID    1     11      1       1.      1.
DVGRID    1      9      1       1.      1.
$
$  Motion for maintaining tangency between the fillet and straight
$  lines
$DVGRID DVID    GRIDID  CPID    COEFF   N1      N2      N3
DVGRID    2     59                      1.          1.         1.
DVGRID    2     59                      1.          1.         1.
DVGRID    2     58                      1.          1.         1.
DVGRID    2     57                      1.          1.         1.
DVGRID    2     56                      1.          1.         1.
DVGRID    2     14                      1.          1.         1.
DVGRID    2     13                      1.          1.         1.
DVGRID    2     12                      1.          1.         1.
DVGRID    2     11                      1.          1.         1.
DVGRID    2     9                       1.          1.         1.
$
$  Assign motion to three grids defining the cylindrical coordinate
system
$
DVGRID    1      1111                   1.          1.         1.
DVGRID    1      1112                   1.          1.         1.
DVGRID    1      1113                   1.          1.         1.

DVGRID    2      1111                   1.          1.         1.
DVGRID    2      1112                   1.          1.         1.
DVGRID    2      1113                   1.          1.         1.
```

Shape Optimization

3 – Geometry Boundary Shape: Example (Input File / Results)

```
$
PARAM, POST, -1
PARAM, DESPCH, 1
$
$  The following has been provided for defining analysis model and
partial
$  Design model.  No user input is required during the workshop.
$
PARAM         AUTOSPC     YES
$
$  GRID          Bulk Data Entry Section for Analysis Model
$ Boundary Conditions for Analysis Model
SPC     1       36      123456  0.0
SPC     1       37      123456  0.0
SPC     1       38      123456  0.0
SPC     1       39      123456  0.0
SPC     1       61      123456  0.0
SPC     1       62      123456  0.0
SPC     1       34      123456  0.0
$
$ Loading Conditions for Analysis Model
$
FORCE   2       7       0       1000.         -1.
FORCE   2       84      0       1000.         -1.
FORCE   2       4       0       1000.         -1.
FORCE   2       78      0       1000.         -1.
FORCE   2       1       0       1000.         -1.
PSHELL  1       1       1.
MAT1    1       2.+8            .3      1.
$
$ Rest of Design Model Definition Section
$
DRESP1  100     WEIGHT  WEIGHT
DRESP1  101     VONMIS  STRESS PSHELL                        9
               1
DCONSTR        1               101                           1.75e4
$
ENDDATA
```

Stress contour plot on the optimized fillet:

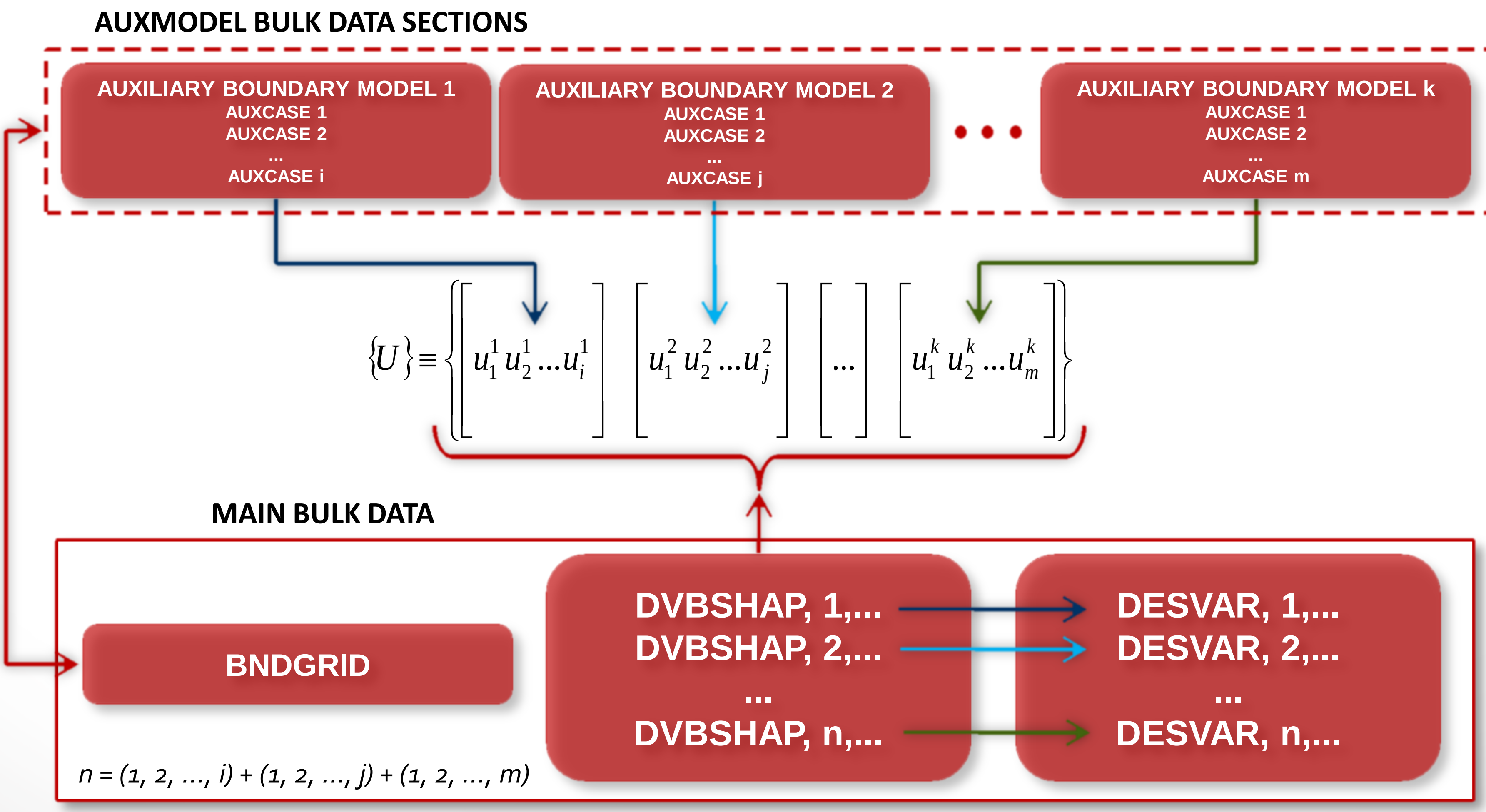


Shape Optimization
4 – Analytic Boundary Shape

- **The auxiliary boundary model is internal to the input file containing the primary model**
  - Each boundary model starts with a **BEGIN BULK AUXMODEL = N** command
  - The **GRID** entries of the primary model are shared
- **The displacement vector resulting from analysis of the auxiliary boundary model denotes shape variation over the boundary (boundary basis vector)**
- **The boundary shape variation is interpolated automatically to the interior grids and this forms the basis vector**
- **BNDGRID Bulk Data Entries are used to identify the GRID IDs and component numbers at the boundary for which motion is:**
  - prescribed – either fixed (by boundary condition)
  - or enforced (by solution of auxiliary boundary model)
- **DVBSHAP Bulk Data Entry connects a DESVAR with a column number of the displacement vector corresponding to a particular auxiliary boundary model**
- **Since shape basis vectors are updated on every design cycle there is little chance of mesh distortion for large shape changes**

Shape Optimization

4 – Analytic Boundary Shape (Graphical Representation)



Shape Optimization

4 – Analytic Boundary Shape – Input File Structure

CASE CONTROL SECTION

CASE CONTROL DECK for the structure
Use SUBCASEs to define the analysis to be considered in the optimization run

AUXCASE

AUXMODEL = 1
Case control structure as in a conventional analysis
(Use SUBCASEs to describe load conditions)

AUXMODEL = 2
Case control structure as in a conventional analysis
(Use SUBCASEs to describe load conditions)

• • •

AUXMODEL = n
Case control structure as in a conventional analysis
(Use SUBCASEs to describe load conditions)

BULK DATA SECTION

Analysis Model

Design Model
BNDGRID
DESVAR
DVBSHAP
DRESPi
DCONSTR
DOPTPRM
...

BEGIN AUXMODEL = 1

Complete input for the auxiliary model 1

BEGIN AUXMODEL = 2

Complete input for the auxiliary model 2

• • •

BEGIN AUXMODEL = n

Complete input for the auxiliary model n

Shape Optimization

4 – Analytic Boundary Shape – Case Control entries

- **Delimits Case Control Commands for an Auxiliary Model in SOL 200**

Format:

AUXCASE

Examples:

AUXCAS

AUXC

Remarks:

1. AUXCASE indicates the beginning of Case Control commands for an auxiliary model. AUXCASE must follow the primary model Case Control commands.
2. All Case Control commands following this entry are applicable until the next AUXCASE or BEGIN BULK command. Commands from preceding Case Control Sections are ignored.
3. Each auxiliary model Case Control must be delimited with the AUXCASE command.
4. The AUXMODEL command is used to associate the auxiliary model Case Control with a particular auxiliary model.

- **References an auxiliary model for generation of boundary shapes in shape optimization**

Format:

AUXMODEL = n

Examples:

AUXMODEL=4
AUXM = 4

Describer Meaning

n	Auxiliary model identification number (Integer > 0)

Remarks:

1. AUXMODEL references a particular auxiliary model for analysis and may only be specified in the auxiliary model Case Control Section.
2. See the BEGIN BULK command for the Bulk Data definition of an auxiliary model.

Shape Optimization

4 – Analytic Boundary Shape – DVBSHAP entry

- **Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model**

Format:

1	2	3	4	5	6	7	8	9	10
DVBSHAP	DVID	AUXMOD	COL1	SF1	COL2	SF2	COL3	SF3	

Example:

DVBSHAP	4	1	1	1.6					

Field	Contents
DVID	Design variable identification number of a DESVAR entry. (Integer > 0)
AUXMOD	Auxiliary model identification number. (Integer > 0)
COLi	Load sequence identification number from AUXMODEL Case Control command. (Integer > 0)
SFi	Scaling factor for load sequence identification number. (Real; Default = 1.0)

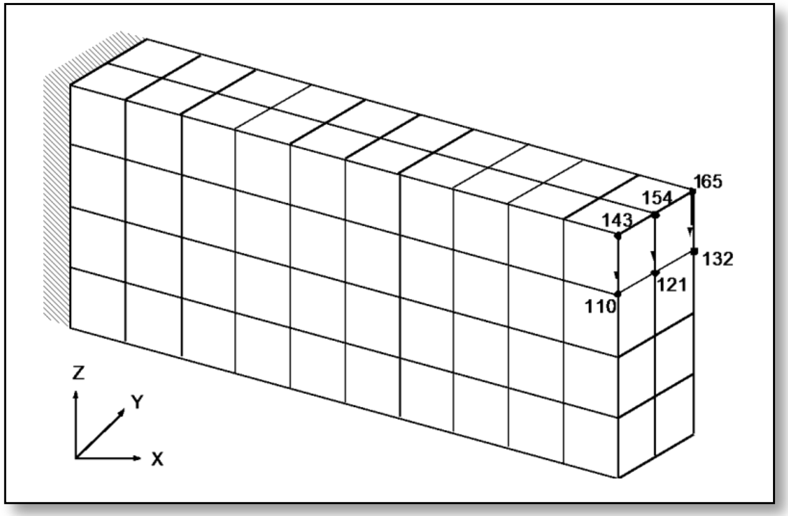
Field	**Contents**
DVID	Design variable identification number of a DESVAR entry (Integer > 0)
AUXMOD	Auxiliary model identification number (Integer > 0)
COLi	Load sequence identification number f rom AUXMODEL Case Control command (Integer > 0)
SFi	Scaling factor for load sequence identification number (Real; Default = 1.0)

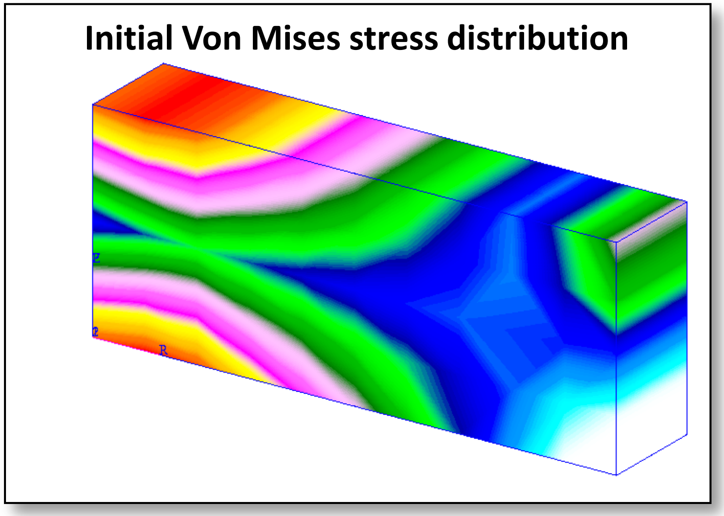
- **Remarks:**
  1. Design variable DVID must be defined on a DESVAR entry.
  2. Multiple references to the same DVID and/or COLi will result in the vector addition of the referenced boundary shape vectors.
  3. Multiple DVBSHAP entries may be specified.

Shape Optimization

4 – Analytic Boundary Shape – Example

- **Consider the following solid cantilever model, with tip loading as shown:**

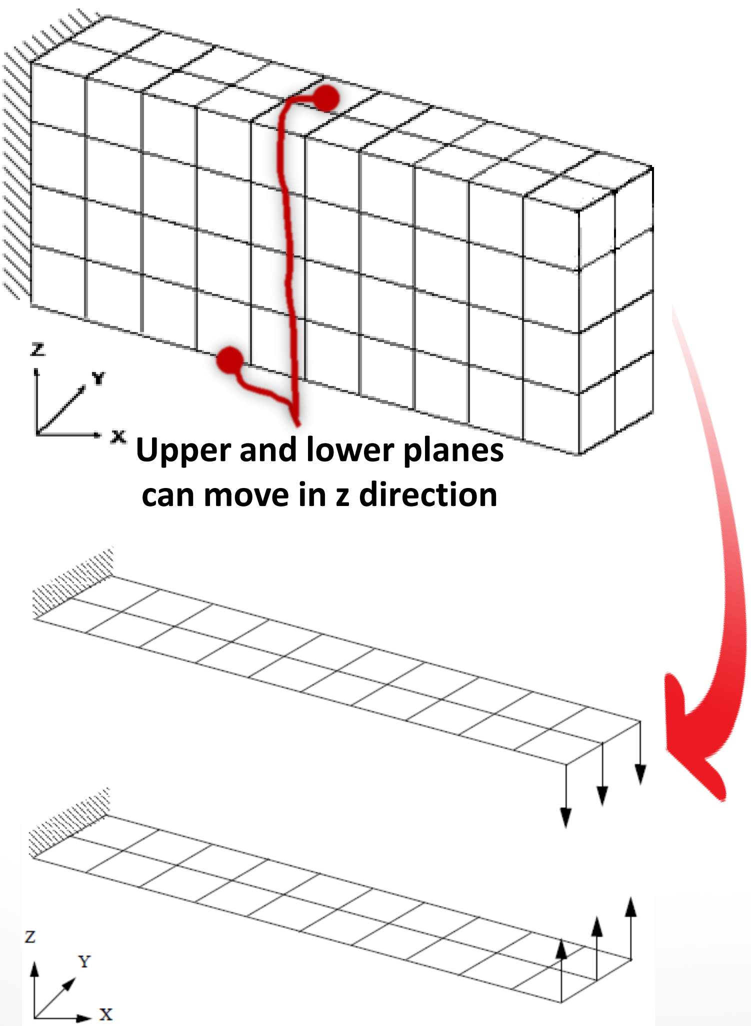




- **For example let us change the shape of the structure to minimize mass subject to constraints on Von Mises stresses.**
- **To change the shape we will modify the upper and lower planes of the cantilever.**

Shape Optimization

4 – Analytic Boundary Shape – Example (Auxiliary Boundary Model)

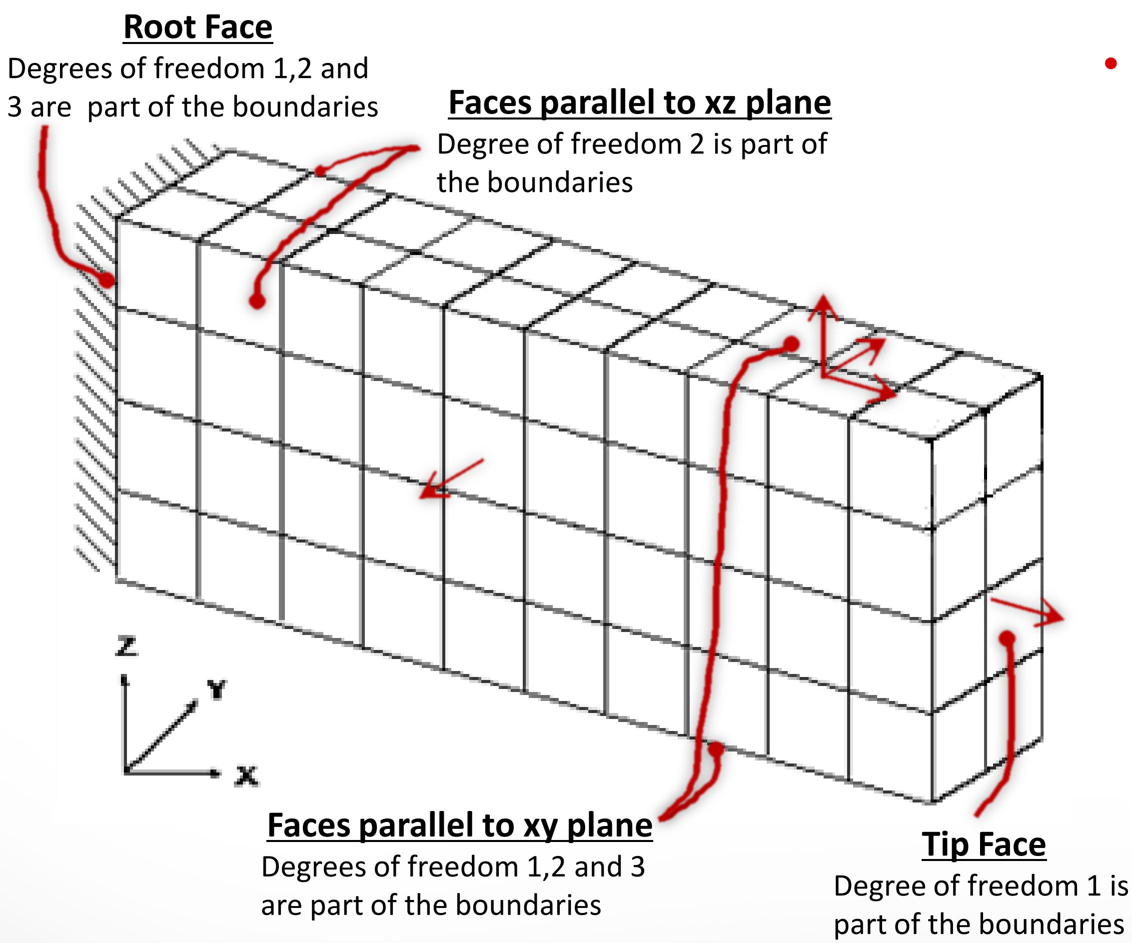


- **The auxiliary boundary model consist of two cantilevered 2D structures that are separately loaded at tip edges**
  - Two separate loading conditions are considered
- **This auxiliary boundary model will produce expected deformations over the primary model boundaries.**
  - These deformations are then interpolated to the interior.
  - The resulting boundary + interior motion will yield a shape basis vector.

$$[T] = [1.0 \cdot U_{SUBCASE200} \quad 1.0 \cdot U_{SUBCASE300}] \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix}$$

Shape Optimization

4 – Analytic Boundary Shape – Example (Boundary definition)



- **The auxiliary boundary model are defined for the faces of the beam parallel to the xy plane**
  - All the other degrees of freedom defined as boundary that are not part of the auxiliary model will be considered as fixed in the relative shape vector
  - All the remaining degrees of freedom (not defined as boundary) will be moved by interpolation

Shape Optimization

4 – Analytic Boundary Shape – Example (Modeling Input)

```
TIME 600      $
SOL  200      $
CEND
TITLE = CANTILEVERED BEAM - HEXA **** D20
 DESOBJ = 15
 DESSUB = 100
SUBCASE 100
 ANALYSIS = STATICS
 SPC     = 1
 LOAD    = 1
 DISPLACEMENT = ALL
OUTPUT(PLOT)
SET 1 = ALL
VIEW 90.0,0.0,90.0
PLOT SET 1
$SET 2 = ALL
$VIEW 34.0, 24.0, 0.0
$PLOT SET 2
$PLOT STATIC DEFORMATION SET 2
AUXCASE
 TITLE = AUXILIARY MODEL 1
 AUXMODEL = 1
 SUBCASE 200
  SPC   = 200
  LOAD  = 220
  LABEL = UPPER
 SUBCASE 300
  SPC   = 300
  LOAD  = 330
  LABEL = LOWER
BEGIN BULK
$
```

```
$------------------------------------------------------------------------
$ ANALYSIS MODEL:
$------------------------------------------------------------------------
$
PARAM   AUTOSPC YES
PARAM   POST    0
PARAM   GRDPNT  0
PARAM   MAXRATIO1.0E+8
$
CORD2S*                2               0               0.0             0.0+1A     2
*1A     2            0.0             0.0               0.0        1000.000+1B     2
*1B     2       1000.000             0.0               0.0             +1C     2
*1C     2
CORD2C*                1               0               0.0             0.0+1A     1
*1A     1            0.0             0.0               0.0        1000.000+1B     1
*1B     1       1000.000             0.0               0.0             +1C     1
*1C     1
GRID           1       0     0.0     0.0     0.0       0
   .           .       .       .       .       .       .
   .           .       .       .       .       .       .
   .           .       .       .       .       .       .
GRID         165       0  10.000   2.000   4.000       0
$GRDSET                                                    456
CHEXA          1       1       1       2      13      12      34      35+EA     1
+EA     1     46      45
     .         .       .       .       .       .       .       .       .
     .         .       .       .       .       .       .       .       .
     .         .       .       .       .       .       .       .       .
CHEXA         80       1     120     121     132     131     153     154+EA    80
+EA    80    165     164
MAT1*                  1      2.0680E+05                  0.28999999166+MA     1
*MA     1     1.00000000 1.169999996E-05                             +MB     1
*MB     1     1500000.00      1500000.00        68000.00             +MC     1
*MC     1
$
```

AUXCASE Section must be defined after any other part of the Case Control Deck relative to the main structure

Shape Optimization

4 – Analytic Boundary Shape – Example (Modeling Input)

```
PSOLID          1        1        0        0        0        0
SPC             1        1   123456      0.0
SPC             1       12   123456      0.0
SPC             1       23   123456      0.0
SPC             1       34   123456      0.0
SPC             1       45   123456      0.0
SPC             1       56   123456      0.0
SPC             1       67   123456      0.0
SPC             1       78   123456      0.0
SPC             1       89   123456      0.0
SPC             1      100   123456      0.0
SPC             1      111   123456      0.0
SPC             1      122   123456      0.0
SPC             1      133   123456      0.0
SPC             1      144   123456      0.0
SPC             1      155   123456      0.0
SPC1     1       456      1        THRU     165
FORCE           1      143        0      0.5      0.0      0.0    -50.0
FORCE           1      154        0      1.0      0.0      0.0    -50.0
FORCE           1      165        0      0.5      0.0      0.0    -50.0
$
$-----------------------------------------------------------------------
-
$ DESIGN MODEL:
$-----------------------------------------------------------------------
$
PARAM,DESPCH,1
PARAM,NASPRT,1
$
$DESVAR,ID,     LABEL,  XINIT,  XLB,     XUB,     DELXV
DESVAR          1 UPPER        1.0      .00     7.00      0.4
DESVAR          2 LOWER        1.0      .00     7.00      0.4
$$DVBSHAP,DVID,  AUXMID, COL1,    SF1,     COL2,    SF2,      ...
DVBSHAP 1       1       1       1.0
DVBSHAP 2       1       2       1.0
$
```

Simmetry is imposed

```
$DLINK, ID,     DDVID,  CO,     CMULT,  IDV1,   C1,     IDV2,   C2,     +
$+,     IDV3,   C3,     ...
DLINK   1       2               1.0     1       1.0
$
$ BOUNDARY CONDITIONS FOR SHAPE INTERPOLATIONS:
$
$ ---TOP SURFACE:
$
BNDGRID,C,      GP1,    GP2,    GP3,    GP4,    GP5,    GP6,    GP7,    +
$+,     GP8,    ...
BNDGRID 123     133     134     135     136     137     138     139
        140     141     142     143     144     145     146     147
        148     149     150     151     152     153     154     155
        156     157     158     159     160     161     162     163
        164     165
$
$ ---BOTTOM SURFACE:
BNDGRID 123     1       2       3       4       5       6       7
        8       9       10      11      12      13      14      15
        16      17      18      19      20      21      22      23
        24      25      26      27      28      29      30      31
        32      33
$
$ ---EXTERIOR SURFACES - INTERPOLATION IN X&Z DIRECTION ONLY:
BNDGRID 2       34      35      36      37      38      39      40
        41      42      43      44
BNDGRID 2       56      57      58      59      60      61      62
        63      64      65      66
BNDGRID 2       67      68      69      70      71      72      73
        74      75      76      77
BNDGRID 2       89      90      91      92      93      94      95
        96      97      98      99
BNDGRID 2       100     101     102     103     104     105     106
        107     108     109     110
BNDGRID 2       122     123     124     125     126     127     128
        129     130     131     132
```

Auxiliary Boundary Model Subcase

Auxiliary Boundary Model ID

Shape Optimization

4 – Analytic Boundary Shape – Example (Modeling Input)

```
$
$ ---TIP END:
BNDGRID 1       11      22      33      44      55      66
BNDGRID 1       77      88      99      110     121     132
BNDGRID 1       143     154     165
$$ ---FIXED END:
BNDGRID 123     1       12      23      34      56      67      89
        100     122     133     144     155
BNDGRID 1       45      78      111
$
$ FORMULATE WEIGHT-BASED SYNTHETIC RESPONSE:  F = 1.E5*W
DRESP1  1       WEIGHT  WEIGHT
DRESP2  15      WE1000  1
+       DRESP1  1
DEQATN  1       F(A)=100000.*A$
$ CONSTRAINTS ON VON MISES STRESSES:
DRESP1  2       STRESS  STRESS  PSOLID                  13              1
DSCREEN STRESS  -1.0    10
DCONSTR 100     2               200.
$
$ OVERRIDE OF OPTIMIZATION PARAMETERS
DOPTPRM DESMAX  9       P1      1       P2      15
$
$-----------------------------------------------------------------------
$ AUXILIARY BOUNDARY MODEL(S):
$-----------------------------------------------------------------------
$
BEGIN BULK AUXMODEL=1
PARAM,PRGPST,NO
PARAM   MAXRATIO1.0E+8
PARAM,AUTOSPC,YES
$
```

```
$ LOWER SURFACE:
CQUAD4  1000    2       1       2       13      12      0.0
CQUAD4  1001    2       2       3       14      13      0.0
CQUAD4  1002    2       3       4       15      14      0.0
CQUAD4  1003    2       4       5       16      15      0.0
CQUAD4  1004    2       5       6       17      16      0.0
CQUAD4  1005    2       6       7       18      17      0.0
CQUAD4  1006    2       7       8       19      18      0.0
CQUAD4  1007    2       8       9       20      19      0.0
CQUAD4  1008    2       9       10      21      20      0.0
CQUAD4  1009    2       10      11      22      21      0.0
CQUAD4  1010    2       12      13      24      23      0.0
CQUAD4  1011    2       13      14      25      24      0.0
CQUAD4  1012    2       14      15      26      25      0.0
CQUAD4  1013    2       15      16      27      26      0.0
CQUAD4  1014    2       16      17      28      27      0.0
CQUAD4  1015    2       17      18      29      28      0.0
CQUAD4  1016    2       18      19      30      29      0.0
CQUAD4  1017    2       19      20      31      30      0.0
CQUAD4  1018    2       20      21      32      31      0.0
CQUAD4  1019    2       21      22      33      32      0.0
$
$ UPPER SURFACE:
CQUAD4  950     2       133     144     145     134     0.0
CQUAD4  951     2       134     145     146     135     0.0
CQUAD4  952     2       135     146     147     136     0.0
CQUAD4  953     2       136     147     148     137     0.0
CQUAD4  954     2       137     148     149     138     0.0
CQUAD4  955     2       138     149     150     139     0.0
CQUAD4  956     2       139     150     151     140     0.0
CQUAD4  957     2       140     151     152     141     0.0
CQUAD4  958     2       141     152     153     142     0.0
CQUAD4  959     2       142     153     154     143     0.0
CQUAD4  960     2       144     155     156     145     0.0
CQUAD4  961     2       145     156     157     146     0.0
CQUAD4  962     2       146     157     158     147     0.0
CQUAD4  963     2       147     158     159     148     0.0
CQUAD4  964     2       148     159     160     149     0.0
CQUAD4  965     2       149     160     161     150     0.0
CQUAD4  966     2       150     161     162     151     0.0
CQUAD4  967     2       151     162     163     152     0.0
CQUAD4  968     2       152     163     164     153     0.0
CQUAD4  969     2       153     164     165     154     0.0
```

Shape Optimization

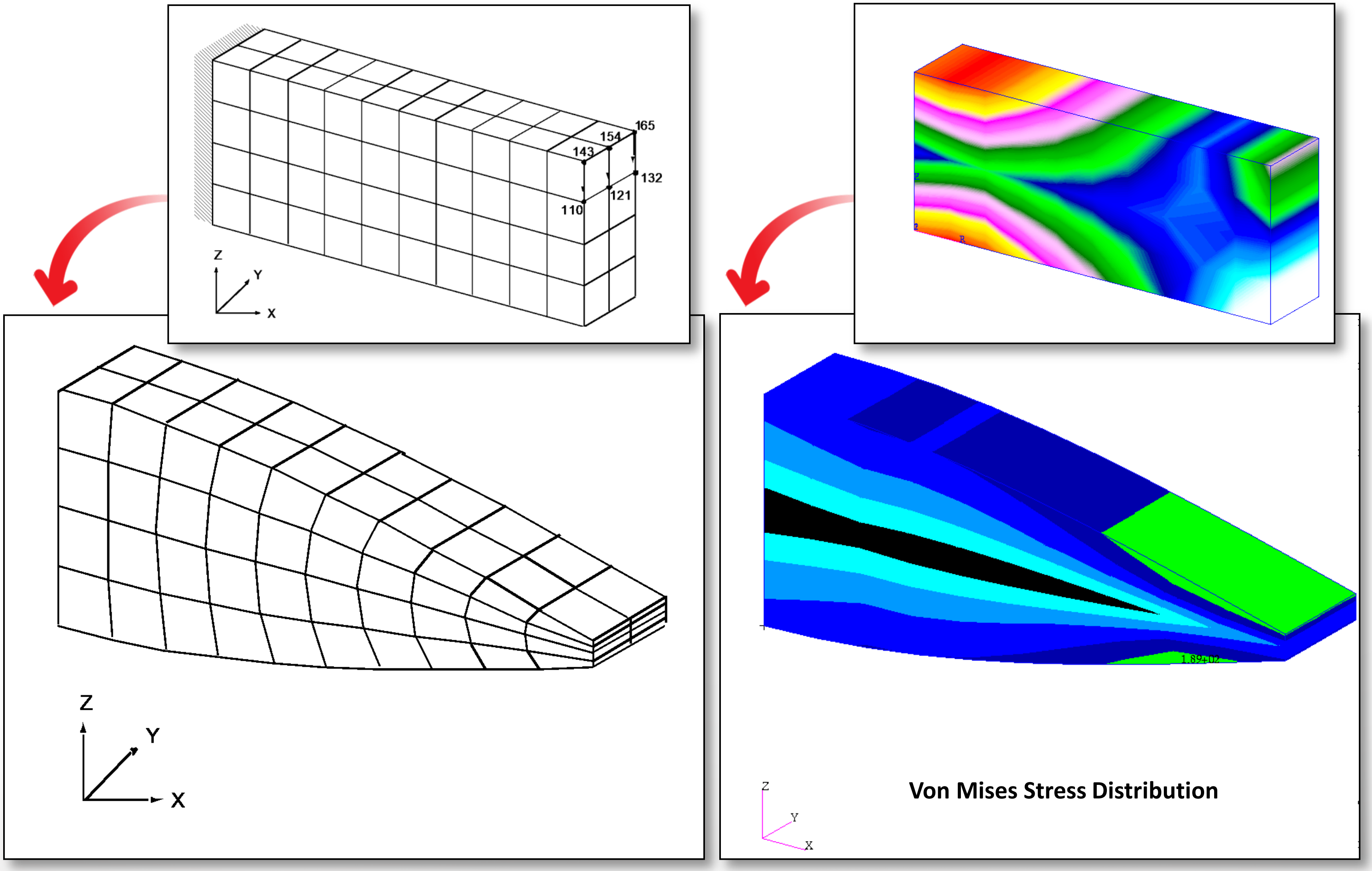
4 – Analytic Boundary Shape – Example (Modeling Input / Results)

```
$
MAT1           11   2.1E+5   0.8E+5      0.3     0.00
PSHELL          2       11     0.20        11
0.0
SPC1          200   123456        1        12        23
SPC1     200      12        11        22        33
SPC1     200      123456    34        THRU      165
SPCD     220      11        3         1.0       22        3         1.0
SPCD     220      33        3         1.0
SPC1     200      3         11        22        33
```

```
$
SPC1            300   123456      133      144      155
SPC1     300      12        143       154       165
SPC1     300      123456    1         THRU      132
SPCD     330      143       3         -1.0      154       3         -1.0
SPCD     330      165       3         -1.0
SPC1     300      3         143       154       165
ENDDATA
```

```
                                 ***************************************************************
                                 S U M M A R Y    O F    D E S I G N    C Y C L E    H I S T O R Y
                                 ***************************************************************
                                           (HARD CONVERGENCE ACHIEVED)
                                           (SOFT CONVERGENCE ACHIEVED)
                                 NUMBER OF FINITE ELEMENT ANALYSES COMPLETED          6
                                 NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS    5
                                        OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY

          ---------------------------------------------------------------------------------------------------------------
                             OBJECTIVE FROM           OBJECTIVE FROM          FRACTIONAL ERROR          MAXIMUM VALUE
               CYCLE          APPROXIMATE                 EXACT                      OF                      OF
               NUMBER         OPTIMIZATION              ANALYSIS               APPROXIMATION             CONSTRAINT
               ---------------------------------------------------------------------------------------------------------------
               INITIAL                                 8.000000E+06                                      -3.827773E-01
                     1          7.401214E+06           7.401214E+06            0.000000E+00              -3.720914E-01
                     2          6.562918E+06           6.562912E+06            9.142283E-07              -3.570341E-01
                     3          5.389287E+06           5.389288E+06           -2.783299E-07              -4.672768E-02
                     4          5.350312E+06           5.350312E+06            0.000000E+00               8.081818E-04
                     5          5.350312E+06           5.350312E+06            0.000000E+00               8.081055E-04
          -----------------------------------------------------------------------------------------------------------------------
           AUXILIARY MODEL 1                                                  MARCH  16, 1994   MSC.Nastran    3/15/94    PAGE    226
                                                         DESIGN VARIABLE HISTORY
        -------------------------------------------------------------------------------------------------------------------------
        INTERNAL |  EXTERNAL   |           |
         DV. ID. |   DV. ID.   |  LABEL    |  INITIAL   :       1       :       2       :       3       :       4       :       5 :
        -------------------------------------------------------------------------------------------------------------------------
              1 |          1   |  UPPER    | 1.0000E+00 :  1.4000E+00 :   1.9600E+00 :  2.7440E+00   :  2.7700E+00 :  2.7700E+00 :
              2 |          2   |  LOWER    | 1.0000E+00 :  1.4000E+00 :   1.9600E+00 :  2.7440E+00   :  2.7700E+00 :  2.7700E+00 :
        -------------------------------------------------------------------------------------------------------------------------
         *** USER INFORMATION MESSAGE 6464 (DOM12E)
             RUN TERMINATED DUE TO HARD CONVERGENCE TO AN OPTIMUM AT CYCLE NUMBER =        5.
```

Shape Optimization
4 – Analytic Boundary Shape – Example (Final Shape)



Shape Optimization
Summary of the shape basis vector generation methods

- **Manual grid variation**
  - Based entirely on DVGRID entries
  - General, yet tedious and prone to input error
- **Direct input of shapes**
  - Based on external auxiliary models
  - DBLOCATE {U}'s, use to form [T]
  - Entries: DBLOCATE, DVSHAP
- **Geometric boundary shapes (GMBS)**
  - DVGRIDs specified over boundaries
  - Code interpolates interior grid motion $\rightarrow$ [T]
  - Efficient coupling with geometric modelers
  - Entries: DVGRID, BNDGRID
- **Analytic boundary shapes (AMBS)**
  - Auxiliary boundary model solutions
  - Code interpolates interior grid motion $\rightarrow$ [T]
  - Interface completely within the MSC/MD Nastran text input environment
  - Entries: AUXCASE, AUXMODEL, BNDGRID, DVBSHAP

Shape Optimization

Guidelines, Recomandations and Limitations

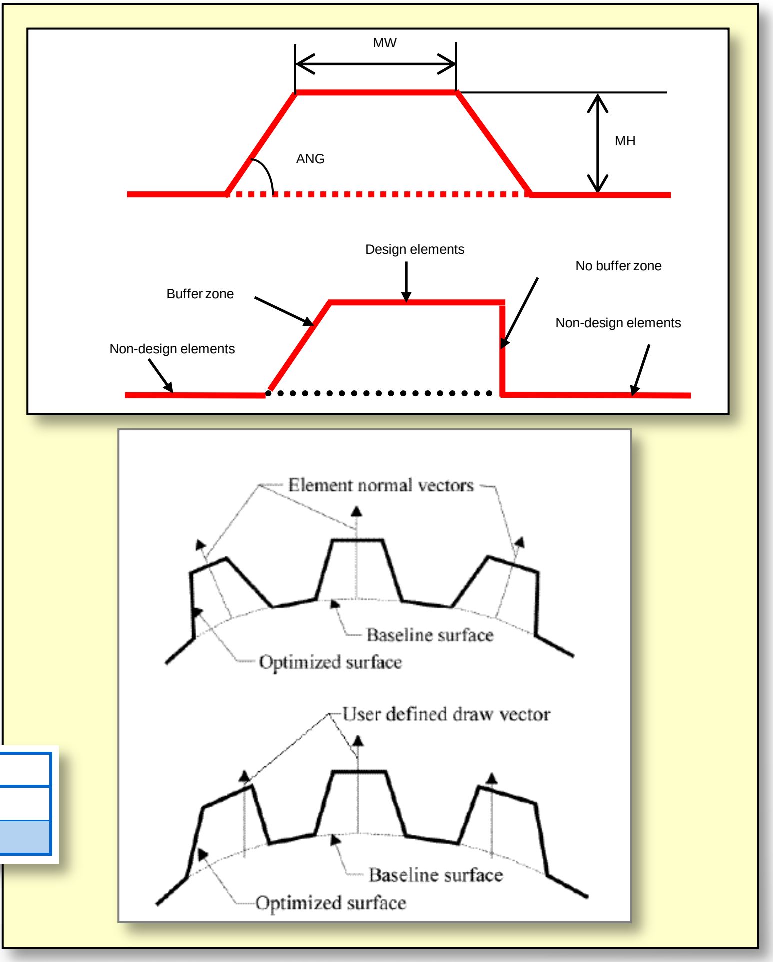
- **Always preview shape basis vectors before starting an optimization job to identify possible modeling errors and to check whether the shape basis vectors satisfy design requirements.**
- **It is recommended to normalize a shape basis vector by its maximum component.**
  - With a normalized basis vector, a user may directly relate a physical parameter (e.g. radius, width) to a shape design variable
  - Therefore, the actual lower and upper bounds can be specified on DESVAR entries
- **Use smaller move limits for a shape design variable (delxv = 0.2 or less)**
- **The new or updated finite element meshes can be shown using Patran.**
  - If other preprocessing software is used, the updated GRID entries in the *.pch file may be used.
- **The general and powerful modeling features, loading and boundary condition input techniques available in MSC Nastran may be used for creating the auxiliary model.**
  - For instance, rigid elements are useful to generate certain basis vectors which change a feature's location but maintain the feature's shape. In addition, temperature or gravity loading may be useful to create special shape changes.
- **Mode shapes generated from a normal modes analysis may be used for basis vectors (Direct Input of Shapes only).**
- **The BNDGRID entries list grids whose displacements will be specified, rather than interpolated. By default, these specified displacements will be zero, unless a nonzero value is supplied via either a DVGRID entry (GMBS) or an auxiliary boundary solution (AMBS).**

A special Shape Optimization Process
Topography optimization

- **Also called Bead or Stamp Optimization**
- **Generate an innovative design proposal for reinforcement bead pattern with a given allowable bead dimension (minimum bead width, maximum bed height, and draw angle).**
- **Topography optimization is particularly powerful for sheet metal parts**
- **It is treated as a special shape optimization and built on SOL 200 shape optimization technology**
  - New algorithms were developed to generate shape design variables and shape basis vectors automatically based on the user's provided bead dimension
- **Many design variables are generated in the topography optimization**
  - The adjoint design sensitivity analysis method and large scale optimizer play key roles in solving topography optimization problems
- **All the analysis supported in SOL 200 can be used**

Topography Optimization
Design Variables for Topography

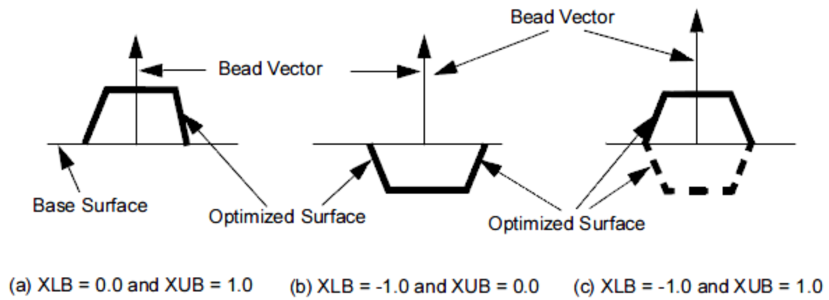
- **The user can provide allowable bead dimension**
- **Grids associated to loads and boundary conditions are skipped**
- **Remove and/or add designable grids**
- **Support PSHELL, PCOMP (PCOMPG), and PSHEAR**
- **Sizing, shape, topometry, topology and topography can be combined in a single job**



BEADVAR	ID	PTYPE	PID	MW	MH	ANG	BF	SKIP	
	"DESVAR"	NORM/XD	YD	ZD	CID	XLB	XUB	DELXV	
	"GRID"	NGSET	DGSET						

Topography Optimization
Design Variables for Topography

- **Bead direction controlled by XLB, XUB**
  - "Above or below surface, or both"



BEADVAR	ID	PTYPE	PID	MW	MH	ANG	BF	SKIP	
	"DESVAR"	NORM/XD	YD	ZD	CID	XLB	XUB	DELXV	
	"GRID"	NGSET	DGSET						

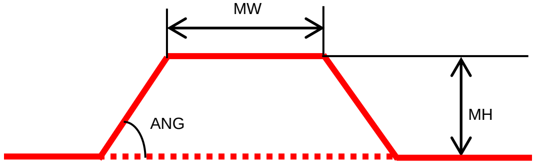
BEADVAR	ID	PTYPE	PID	MW	MH	ANG	BF	SKIP	
	"DESVAR"	NORM/XD	YD	ZD	CID	XLB	XUB	DELXV	
	"GRID"	NGSET	DGSET						

- **Additional Design region (GRID) control**
  - SKIP = boundary condition flag (allow SPC's to move or not move)
    - BC – GRIDs attached to spc/spc1 are not allowed to move
    - LOAD – GRIDs attached discrete loads are not allowed to move
      - FORCE, FORCE1, FORCE2, MOMENT, MOMENT1, MOMENT2, SPCD
    - BOTH – Neither BC nor LOAD GRIDs are alloed to move
    - NONE – GRIDS allowed to move regardless of loads/spcs
  - "GRID"
    - NGSET – GRIDs removed from design region
    - DGSET – GRIDs added to design region

Topography Optimization
Modal Analysis Example

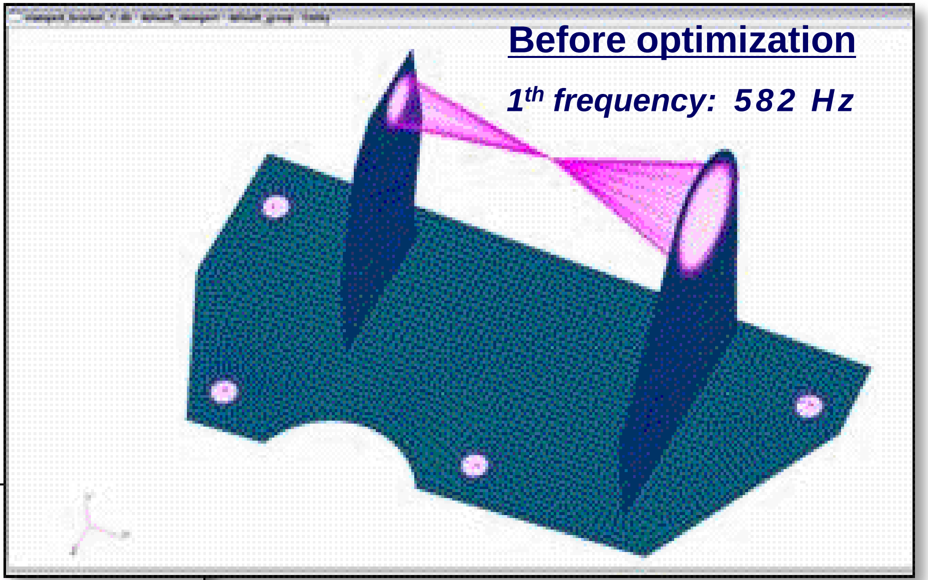
Maximize 1st Frequency

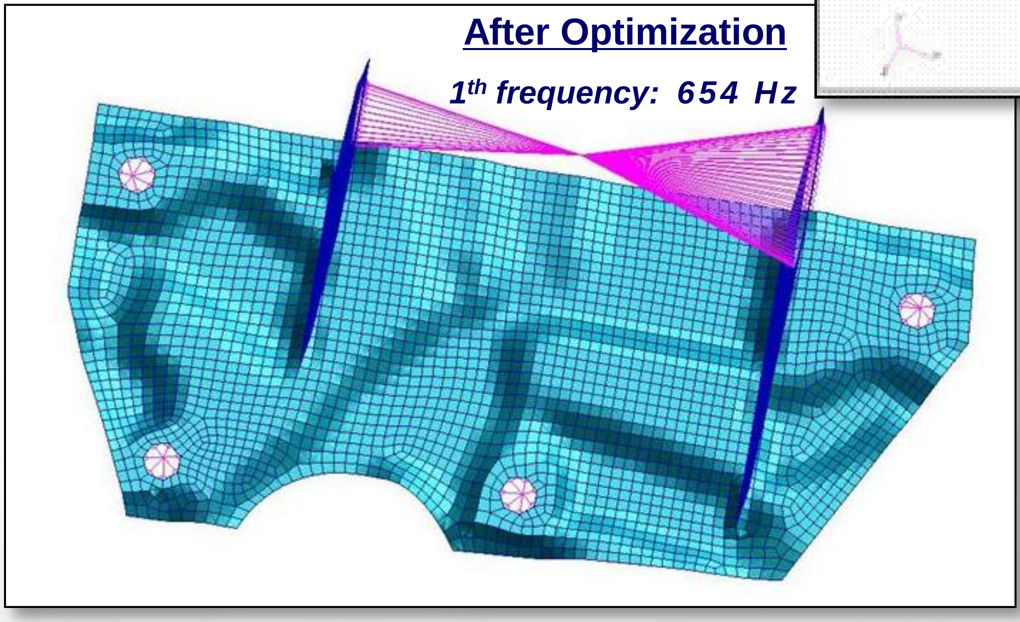
– Design the base only



Minimum width (MW)= 10.0
Maximum height (MH)= 20.0
Draw angle (ANG) = 70.0

Grids describing the holes are fixed



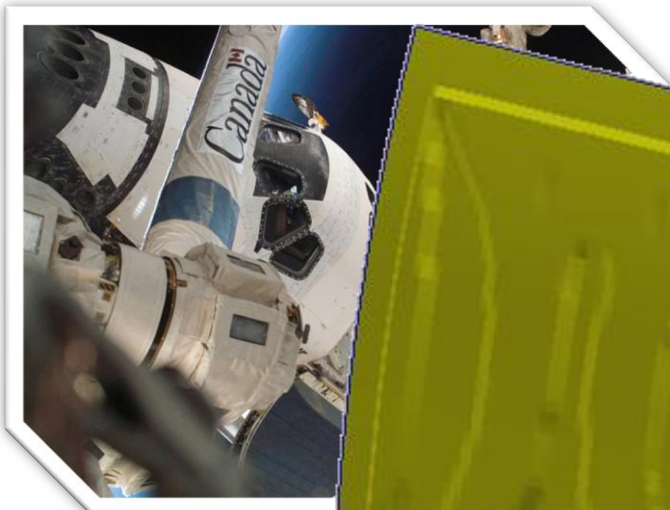


If needed the position of some grid points in the design region can be fixed by:

- Defining the affected region
- Declaring it in the Topology Optimization control entry

Topography Optimization

Real life example



Initial Design

$R=90.$, $\Theta=28.6^{\circ}$

$Z=30.$, $t=.08$

$Wt_{init}=10.78$,

$F1_{init}=16.8Hz$

…ventional Design

…79.5Hz

13 cycles

…e 2 -

…: Min Wt

… F1>35Hz

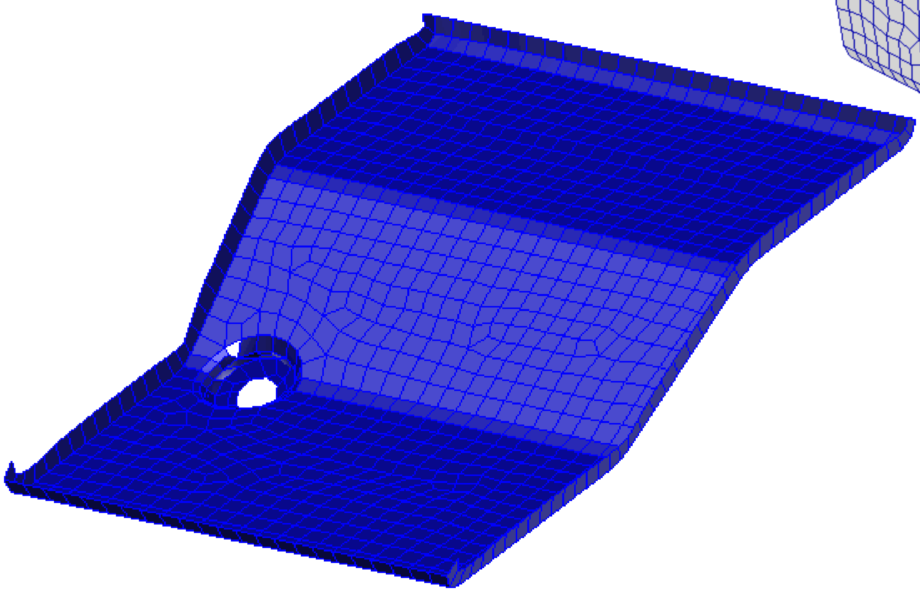
….0 Hz

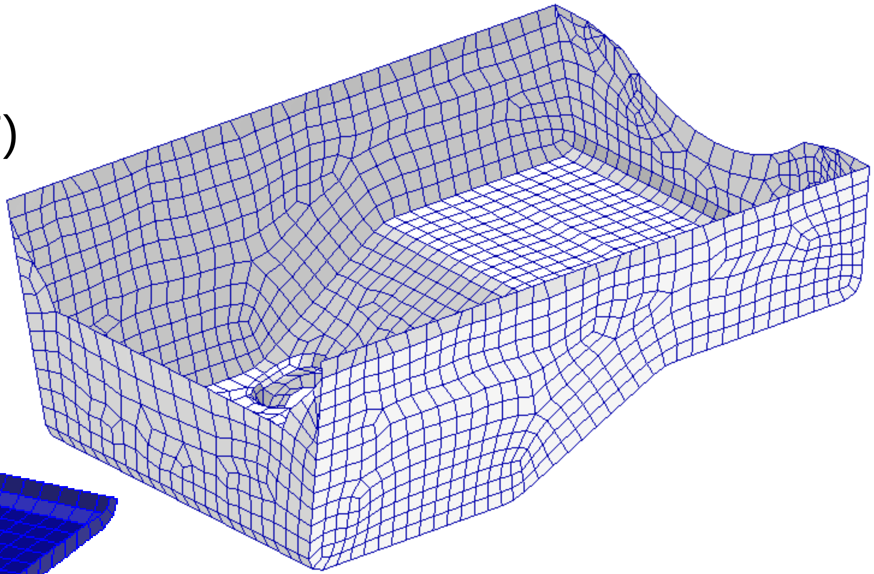
…).81

…es

Topography Optimization
Real life example

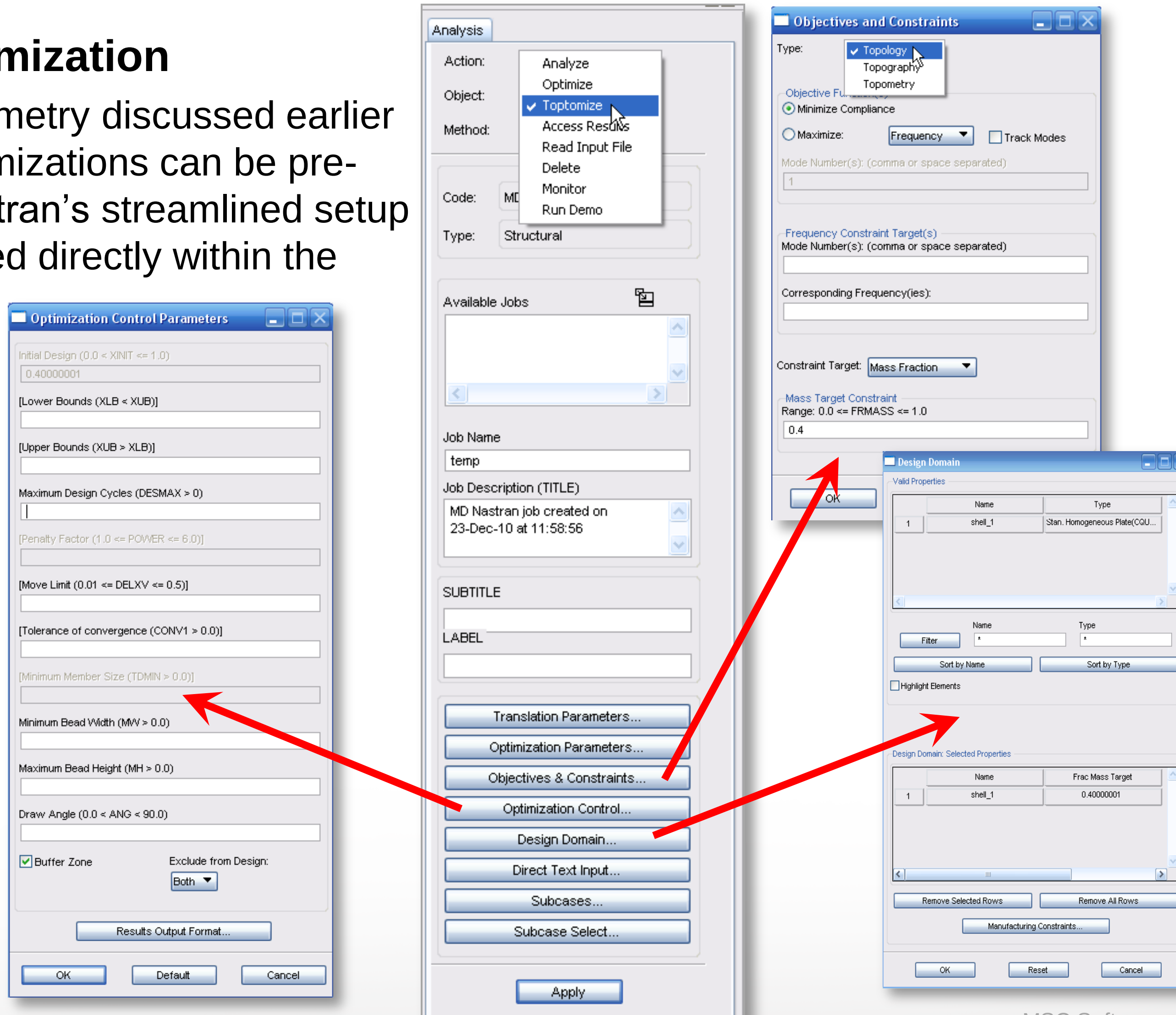
- **togex3.dat**
  - Oil pan, free free
  - Objective: max 1st flex mode (f7)
  - Animate shape change
    - Initial 94 Hz, Final 222 Hz

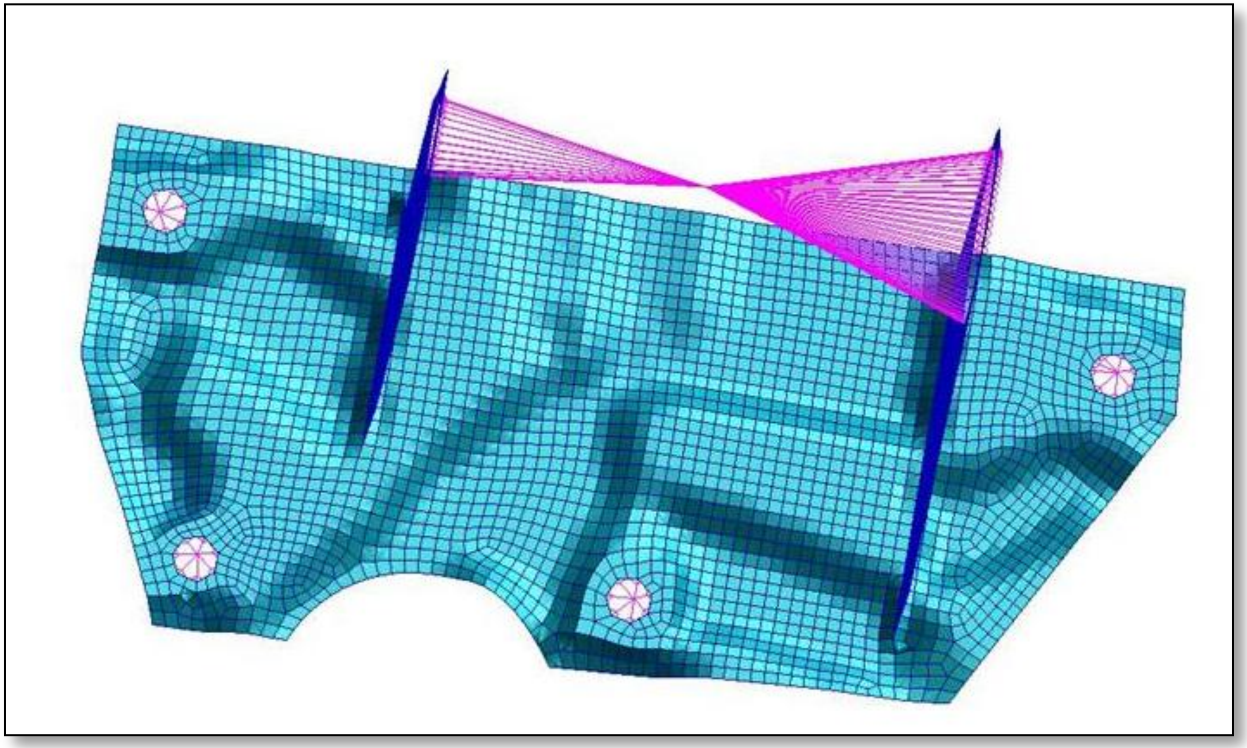




Topography Optimization
GUI

- **Topography Optimization**
  - Similarly to topometry discussed earlier topography optimizations can be pre-processed in Patran's streamlined setup process accessed directly within the 'Analysis' form:





Mauro Linari
Project Manager
+39 06 84404523
mauro.linari@mscsoftware.com

Thanks !

http://www.mscsoftware.com/