
Pro/MECHANICA® Wildfire™ 3.0

Some Revisions from Wildfire 2.0

Ray Ellender, Managing Director, Elite Consulting Ltd

Presented at the Mechanica User Group Meeting 9 July 2008

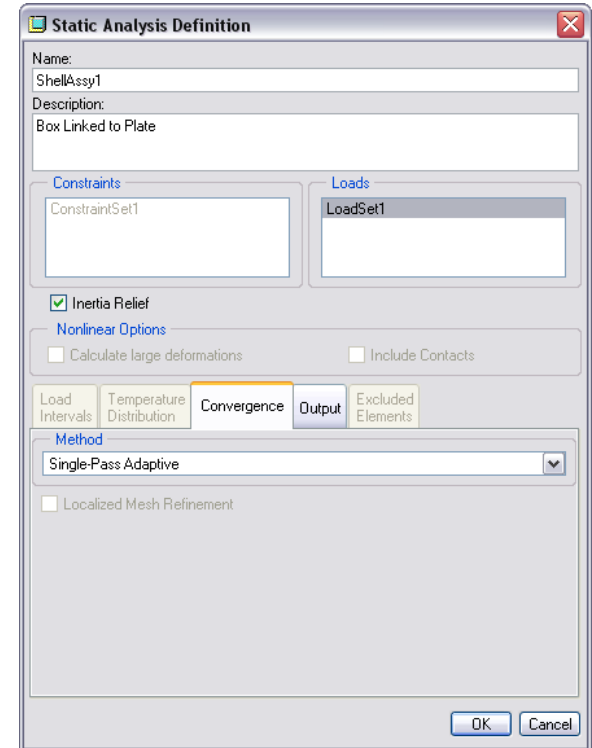
© Elite Consulting Ltd, 2007, 2008

Wildfire 3.0 Highlights

- Inertia Relief
- Weighted Links
- Pro/E Materials
- Volume Heat Loads
- Changes to Contact
- Changes to Sensitivity & Optimisation
- Windows XP 64-bit Support
- Process Guide

Inertia Relief

- Inertia Relief
 - New for Wildfire 3.0
 - Allows a static analysis without applying constraints
 - Uses D'Alembert's principle: applies an acceleration load to balance forces
 - The user must create all the applied loads, so any reaction loads must be calculated and applied to the model
 - Using this option, Mechanical analyzes your model as if it were floating freely in space, without any constraints, but with the loads applied.



Inertia Relief

Inertia Relief

- You can use this option only for a linear static analysis.
- **Note:** You can also use the **Inertia Relief** option to analyze an underconstrained model. However, if the model already has some constraint sets, then Mechanical ignores these constraints during the analysis.
- During an analysis with the **Inertia Relief** option selected, Mechanical internally creates a new Cartesian coordinate system (UCS) and defines a constraint set containing three-point constraints with respect to the newly created UCS. Mechanical also automatically applies body loads that balance your applied loads.
- The three-point constraints affect the displacement solution but not the stress solution.
- An analysis with inertia relief should always run fine, as long as there is non-zero stiffness for all the six degrees of freedom between two bodies.
- Before you run an analysis with inertia relief, Mechanical asks you whether you want error detection to be performed. If you do, Mechanical checks for various modeling conditions including the presence of multiple bodies in the model. If Mechanical encounters more than one body, then it displays a message indicating the number of separate or disjoint bodies that it finds. You can use this information to determine whether parts you thought were connected are truly connected. If you see an unexpected number of bodies, you may want to cancel the analysis or study and correct the assembly.
- For an analysis with Inertia relief ensure that the model does not have more than one disjoint body. If multiple disjoint bodies exist in your model, then the analysis fails with an underconstrained error. To run an inertia relief analysis with multiple disjoint bodies, ensure that all the disjoint bodies are connected in such a way that there is no relative motion between the bodies. If the connections are such that relative motion exists between any two bodies in the model, then the inertia relief analysis fails with an underconstrained error. For example, if two bodies are connected by bolts, then the inertia relief analysis fails if the bolts have no stiffness for some degrees of freedom.

Old Method – 3 Point Constraint

3-Point Constraint – Con Rod

Open con-rod.prt & bring into Mechanica.

3 points on the mid-plane

Set up a 3-point constraint using these points.

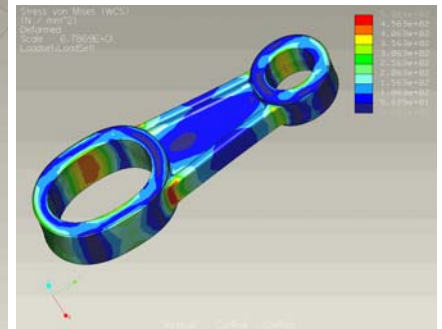
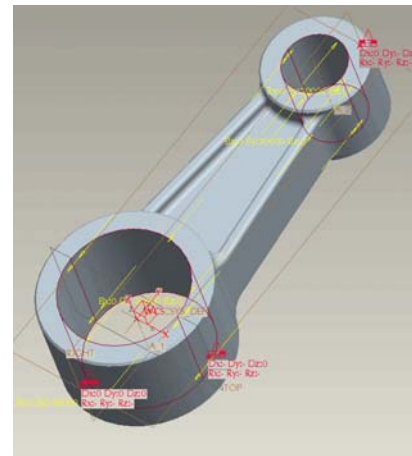
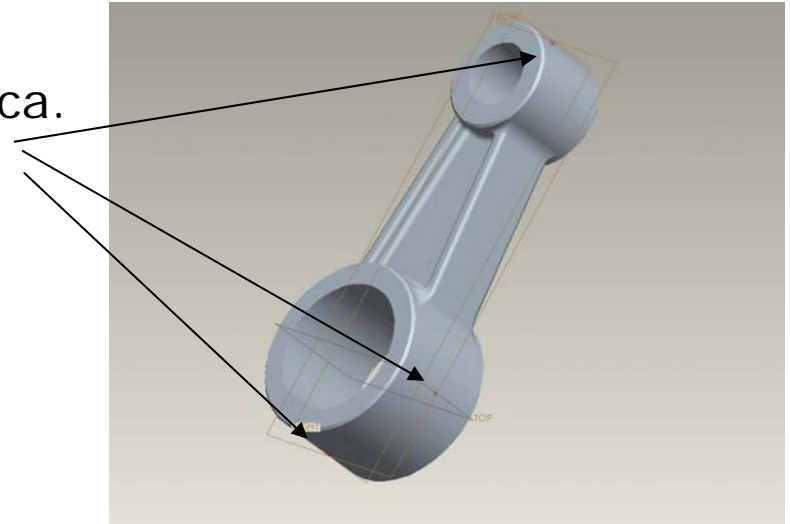
Apply a 20kN tensile load, as a bearing load to the small end

Apply a -20kN reaction load as a bearing load to the big end

Material is Steel

Set-up and run a single pass adaptive analysis called ConRod. (You may get a warning about the point constraints causing singularities, but carry on)

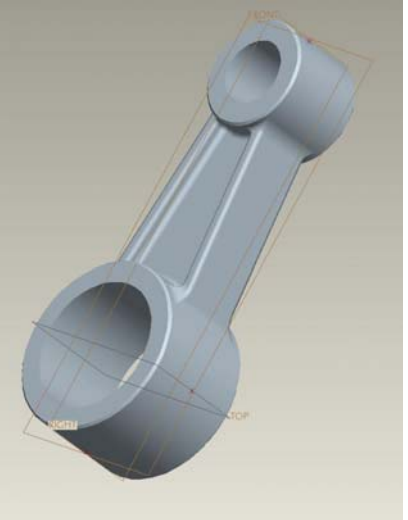
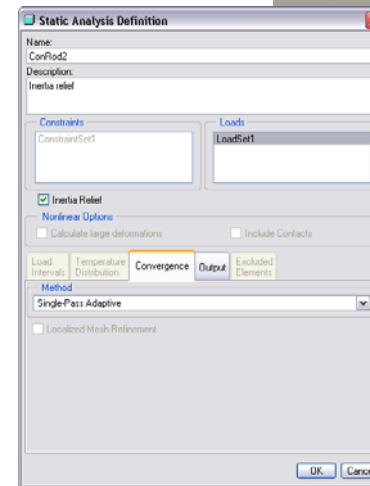
Review the results



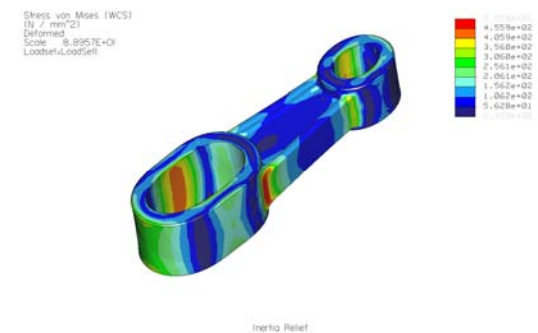
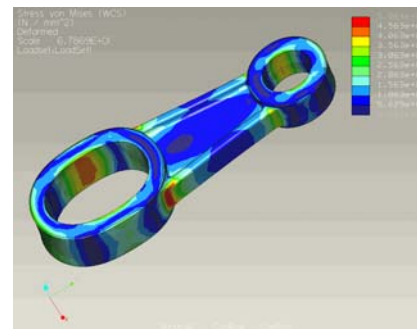
New Method – Inertia Relief

Inertia Relief – Con Rod

Set-up and run a single pass adaptive analysis called ConRod2. Select the Inertia relief tick-box. The constraint set will be greyed out



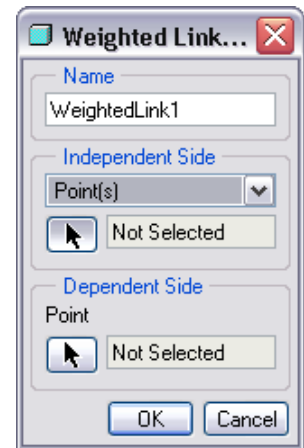
Review the results and compare to the 3-point method



Weighted Links


Weighted Links

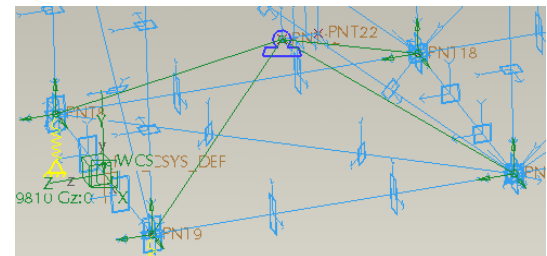
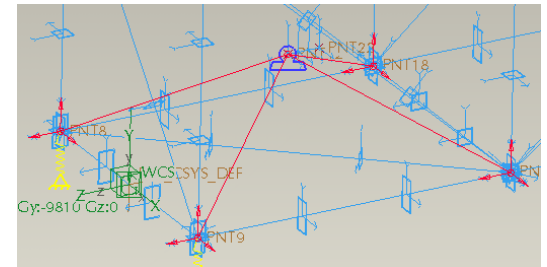
- A weighted link is an interpolation constraint element that takes masses or loads acting at a single source point and distributes them to a collection of target geometric entities. Weighted links have the following characteristics:
 - Weighted links distribute masses or loads in a balanced manner
 - Weighted links can have only one source point that follows the average motion of the target entity group. Thus, the source point is called dependent and the target entities—dependent
- These characteristics make weighted links useful when you need to attach mass idealizations to your model without stiffening the structure
- For the independent side of the link, select any of the following references:
 - Point(s)
 - Edge(s)/Curve(s)
 - Surface(s)
- For the dependent side of the link, select or create a point as a reference entity



Connecting Mass Elements

Connecting Mass Elements – Weighted Links (If Advanced Licence available)

- ❑ In WF2, use light stiff beams or rigid links to connect the a mass element representing the engine mass to the chassis. We will connect the engine mass to the chassis with Point to point Weighted Links
- ❑ In WF3 can use Weighted Links from PNT12 to the four points at the bottom corners of the engine bay
- ❑ Go to Insert>Connections>Weighted Links or click on the Weighted Link Icon 
- ❑ For the 'Dependent Side', select PNT12, the point to which the Engine mass element is attached.
- ❑ For the 'Independent Side', select the 4 points at the corners at the bottom of the engine bay; PNT2, PNT3, PNT11, PNT18
- ❑ Then click OK
- ❑ These points will then show as connected in the model






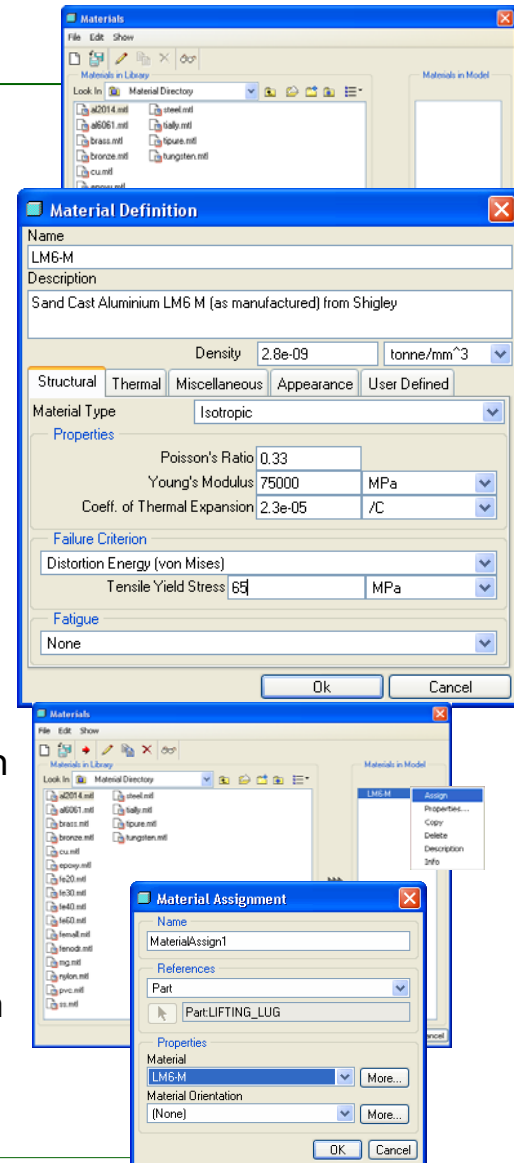
Materials

- ❑ From Wildfire 3.0, materials can be assigned in either Pro/Engineer or Mechanical; the material definitions in Pro/Engineer and Mechanical are exactly the same. Materials assigned to parts in Pro/Engineer are read directly in Mechanical; there is no need to re-assign the material.
- ❑ There is a small material library, which you can add to.
- ❑ You can set up new materials
- ❑ Note : If you have a part has had a material assigned to it in Pro/Engineer and that material does not have E or ν assigned, the run will crash.

Assign a Material

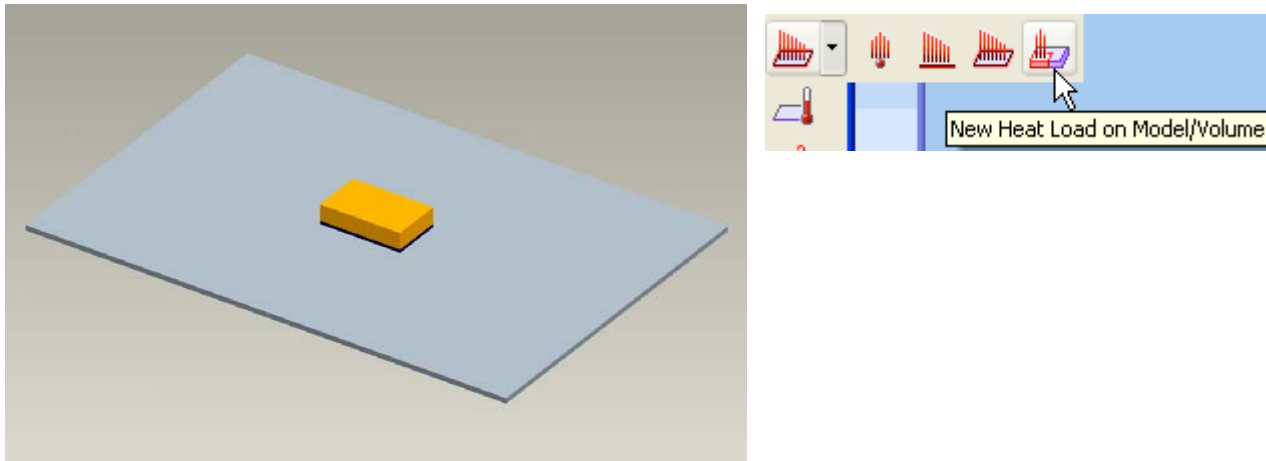
Materials – Define & Assign a Material

- ❑ **If you are in Mechanica**
- ❑ Click on the Define Materials Icon 
- ❑ A Materials Window comes up
- ❑ Move AL2014 across to be in the model, then copy it to make a new material. Give the copy the name of LM6-M and enter the data as follows: -
 - Poisson's Ratio = 0.33
 - Young's Modulus = 75000 MPa
 - CTE = 2.3e-05 /C
 - Density = 2.8e-09 tonne/mm³
 - Failure Criterion
 - Von Mises, Tensile Yield = 65 MPa
- ❑ Assign material by clicking on the Material Assign Icon  in Mechanica and selecting LM6-M and clicking OK. A material icon will appear on the model. 
- ❑ **Alternatively in Pro/Engineer**, go to Edit>Setup>Materials and define the LM6-M material. Then assign it by right-clicking on it.
- ❑ If the material has been assigned in Pro/E, there will be no icon on the Mechanica model, but when the assigned material is selected in the model tree the part will highlight in dark red.



Volumes

- Ability to assign heat loads by volume

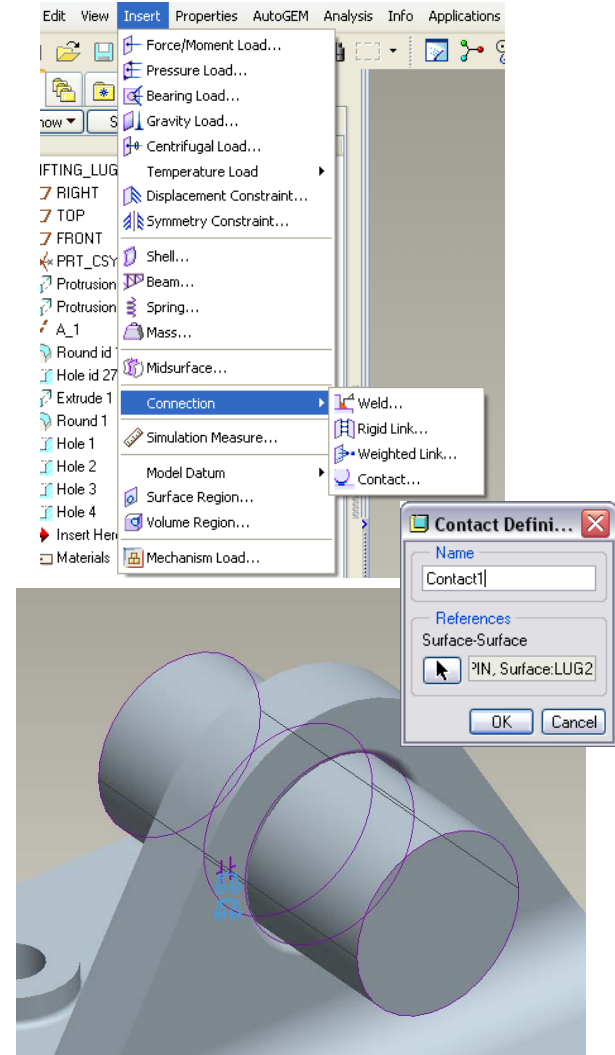


- Ability to assign materials by volume
 - Volume regions in a part can have different properties

Contact Regions

Surface Contact Regions

- ❑ Pro/E splits cylindrical surfaces into 2 halves, A & B
- ❑ For a shaft in a hole in WF2 had to make separate regions, especially if they were at 90o to each other
 - Hole Surf A v Shaft Surf A
 - Hole Surf A v Shaft Surf B
 - Hole Surf B v Shaft Surf A
 - Hole Surf B v Shaft Surf B
- ❑ In WF3 this problem is overcome, just have to select one of the 'half-surfaces' on the shaft and one of the 'half-surfaces' in the hole, and contact on each whole surface is automatically calculated.
- ❑ Select the one of the pin surfaces and one of the inner surfaces of the hole in the lug

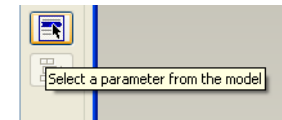
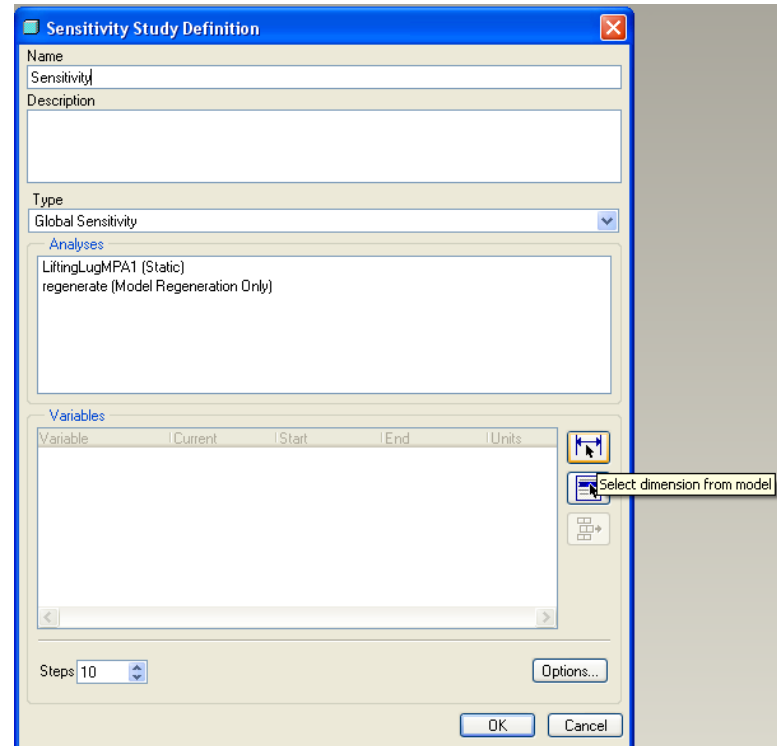


Design Studies

- The last of the “Old Style” UI removed
- Usability improvements to...
 - Optimization definition
 - Sensitivity study definition
 - Design Studies
- No more design variable definition
 - Use dimensions and parameters directly
- Greater control of optimization settings
 - Choose GDP or SQP
- Feasibility studies added

Design Parameters

- Parameters are a way of defining which dimensions can vary and the range permitted
- Parameters can be also be created using Pro/Engineer parameters
- Pro/Engineer relations can be used to ensure design intent



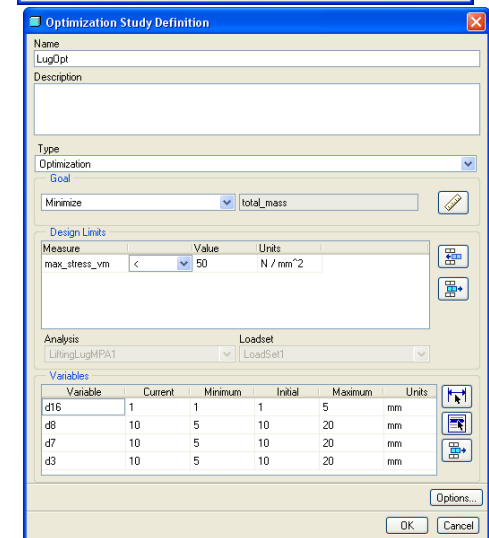
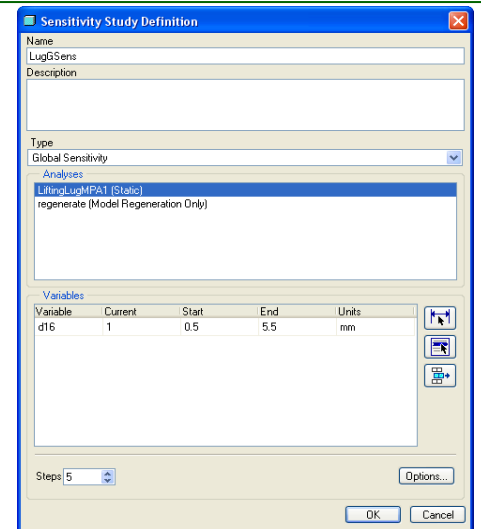
Sensitivity & Optimisation

Global Sensitivity

- Add parameters by selecting directly from Design Study Box
 - Add the bolt boss fillet (d16) by selecting the dimension icon
 - Set the start to 0.5mm and the end to 5.5 mm
 - Set the number of intervals to 5
 - **WARNING – there was a bug in Wildfire 3.0 that does not allow the plotting of any results if there are user defined measures!!!**
 - Check if the model has any measures and delete before running the analysis

Optimisation

- Add parameters by selecting directly from Design Study Box
 - **Note : Have to set up parameters again**
 - Design Limits: max_stress_vm < 50 MPa
 - Select the dimensions that are allowed to change
 - Click on the Options box and set the Optimisation Algorithm to Automatic (you could choose GDP or SQP)



Simulation Advisor, Process Guide, Help & Support

To help you with the process of completing a successful analysis, PTC provide Mechanics Users with the following

1. Help Files
2. Simulation Advisor
3. Process Guide (New for WF3)
4. Technical Support

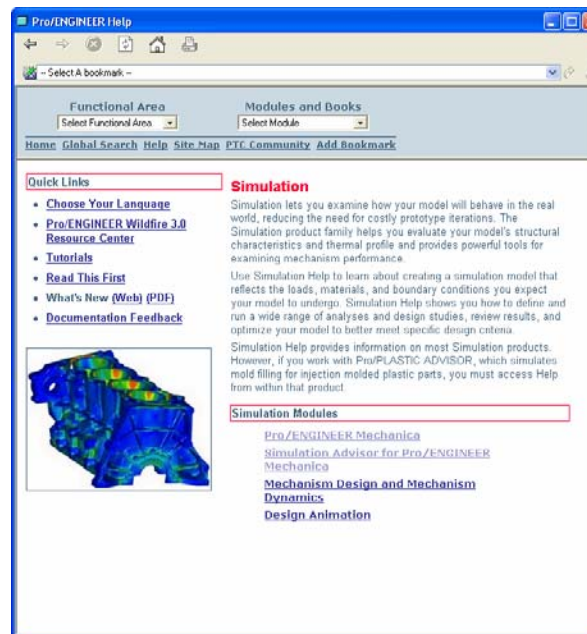
Simulation Advisor (In WF2 & WF3)

This is also accessed via the Help  icon.

On the Help Home page, click on Simulation, then Simulation Advisor.

Then you can go to advice on

- Assemblies,
- Geometry,
- Model,
- Simulation,
- Results
- Optimisation



Process Guide (In WF3)

New for Wildfire 3.0, a Process Guide Wizard is available from within Mechanica, just click on the  icon.

A box to define a new Process Guide session comes up (or if a session already exists a box to choose the existing session). You choose the appropriate template and that brings up the relevant process guide wizard window. This takes you through the various steps in the process. You can undertake the analysis tasks from within the Process Guide and it keeps a record of what you have done, e.g. create loads using the Loads Manager available within the Process Guide.

Standard templates are provided, but you can design your own templates using XML

