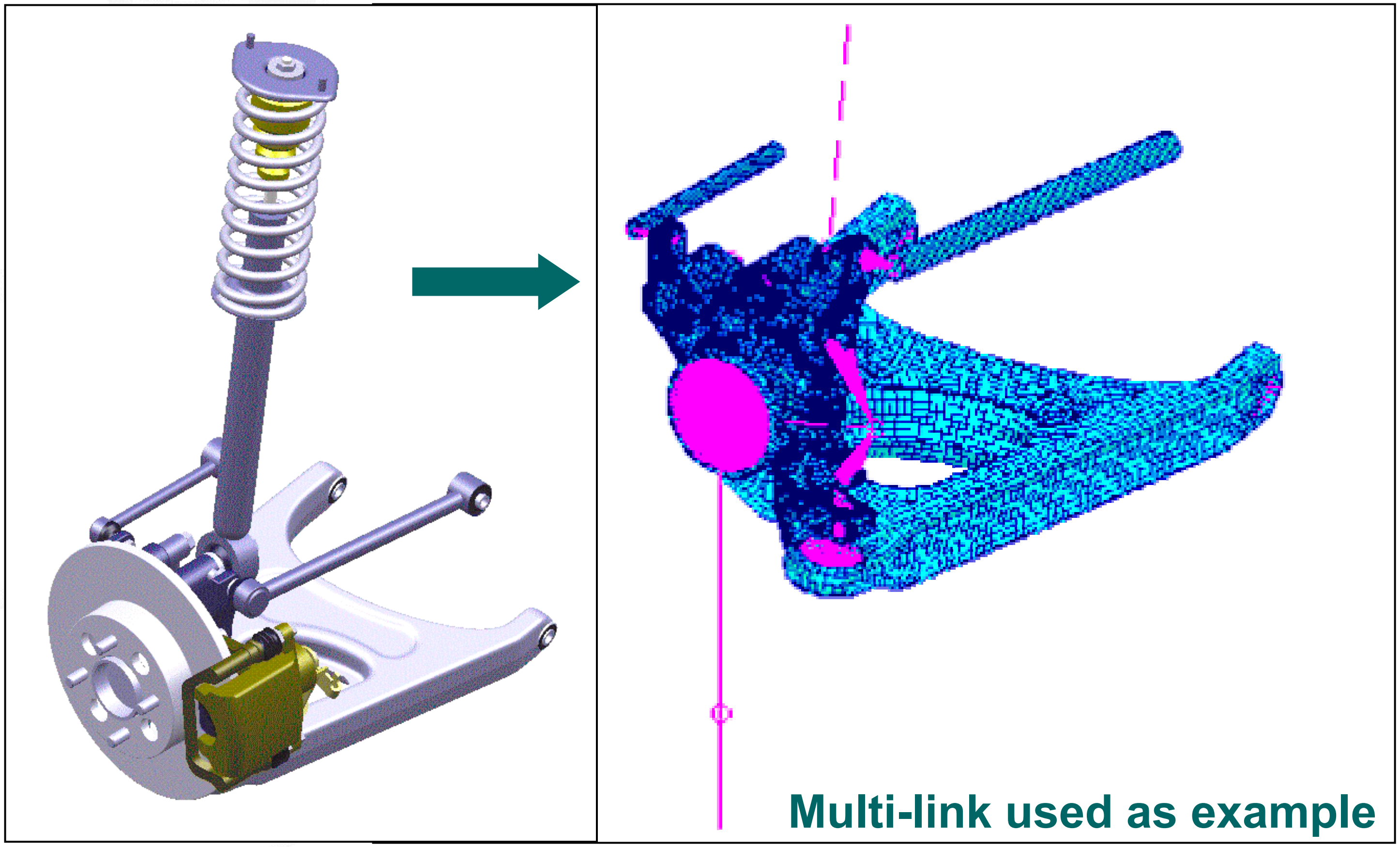
CAE STRUCTURAL APPROACH

Part Physical Geometry

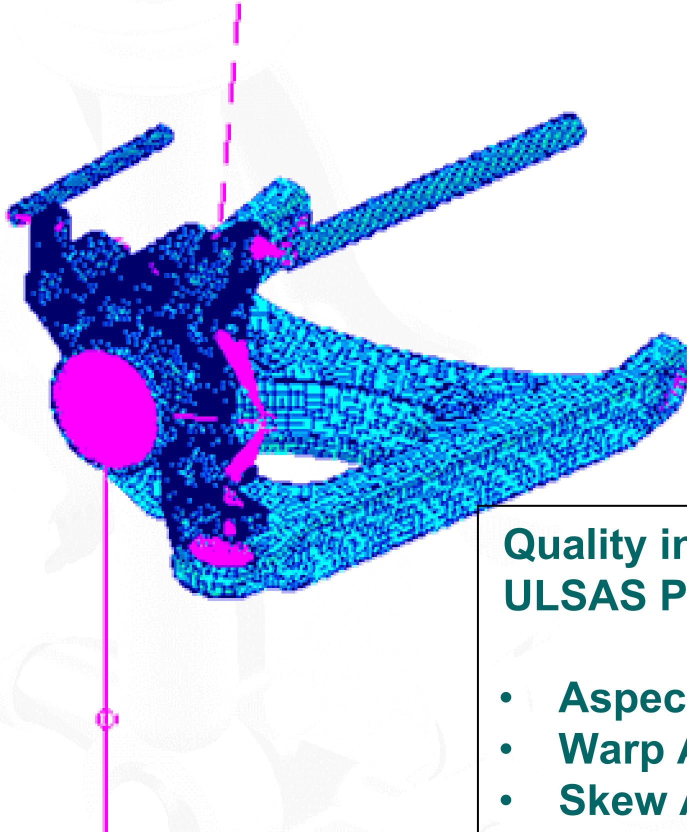


Multi-link used as example

The physical geometry of the parts used to create the finite element model was imported from the CAD environment. This was modified within the FE environment using the many tools available.

CAE STRUCTURAL APPROACH

Finite Elements



Multi-link used as example

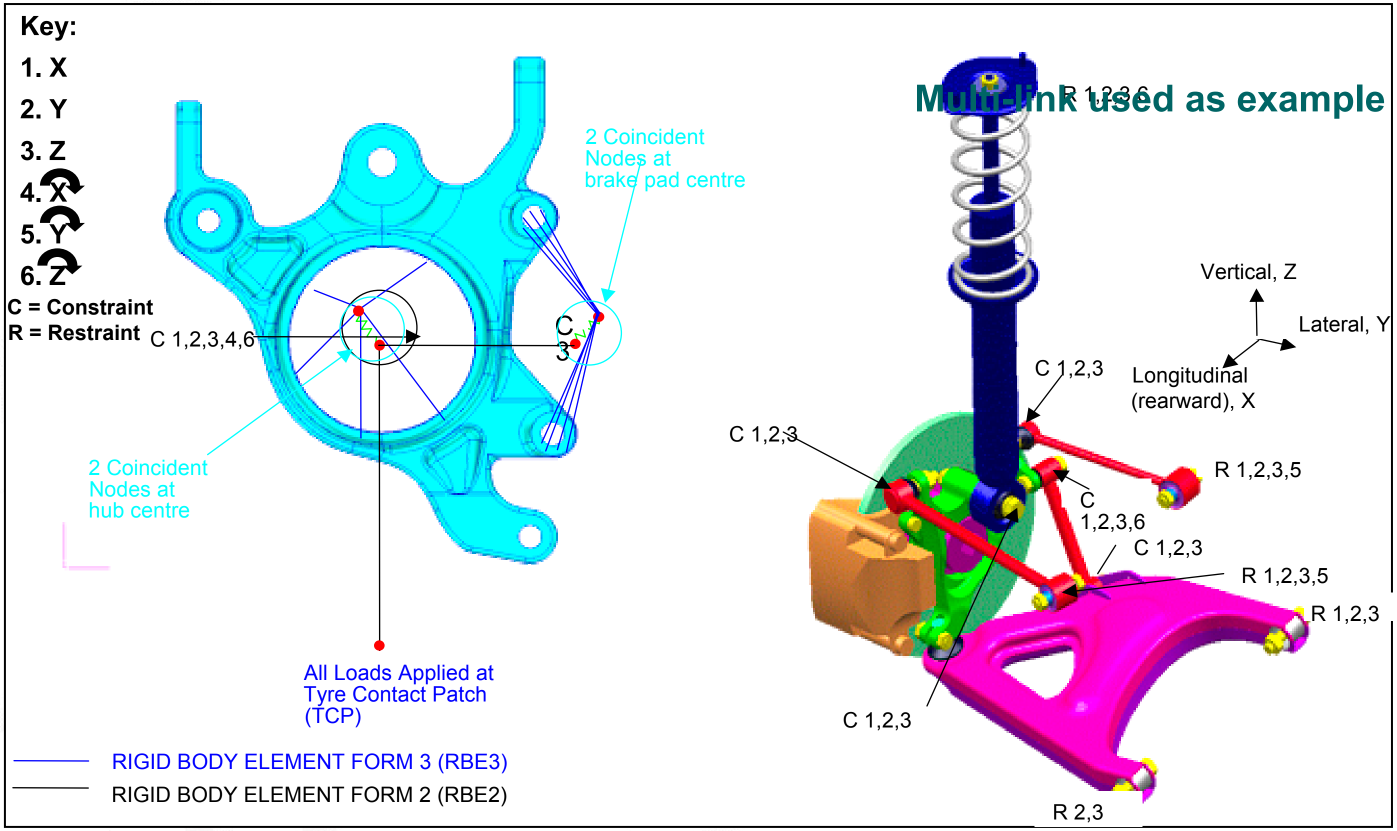
Quality indices adapted throughout the ULSAS Programme for shell elements :

• Aspect Ratio	<	5:1
• Warp Angle	<	7 degrees
• Skew Angle	<	30 degrees
• Taper	>	0.8

An FE mesh was created using the imported CAD geometry. This was undertaken by using either manual or auto meshing techniques. Beam, shell or solid elements are used depending upon the underlying geometry. Once the mesh has been created, it is checked for free edge duplicates and normals. The element's quality is also checked for aspect ratio, warp angle, skew angle, and taper. Typical values for these are:

CAE STRUCTURAL APPROACH

Loads and Boundary Conditions



Restraints, constraints and loads are applied to the FE model using appropriate rigid elements and springs, with the necessary degrees of freedom carefully defined. Restraints are normally RBE3s from a hole to a fixing point, and then a spring to ground. Constraints connect two components using RBE3s from holes to a common point, which is joined using springs. Loads are applied through RBE2s and RBE3s to the structure. NB. RBE3s are defined as the motion at a reference grid point as the weighted average of the motions at a set of other grid points and RBE2s are defined as a rigid body whose independant degrees of freedom are specified at a single grid point and whose dependant degrees of freedom are specified at an arbitrary number of grid points.

CAE STRUCTURAL APPROACH

Materials

Material models are obtained from the FE software database, or else are created explicitly. Linear analysis only requires the elastic modulus and Poisson ratio. A non linear analysis also requires the yield point and a plastic hardening modulus.

Properties

Spring, beam and shell properties are defined for each type of element. Springs require stiffnesses and degrees of freedom, beams require section properties and orientations, and shells require thicknesses.

CAE STRUCTURAL APPROACH

ULSAS Standard Load Cases

Load Case Description (2)	X direction	Y direction	Z direction (1)	Position of force Application
Reverse Curb Strike	- 0.5 g	0	3 g	Tyre contact patch
Lateral Curb Strike 1	0	(-) 1.5 g (based on axle weight)	1g with weight transfer	Wheel rim lower position
Lateral Curb Strike 2	0	(-) 1.5 g (based on xle weight)	1g with no weight transfer	Wheel rim lower position
Vertical Bump	0	0	4 g	Tyre contact patch
Forward Braking (With ABS)	1.1 g	0	1g with no weight transfer	Tyre contact patch
Combined Bump and Cornering	0.316 g at wheel including yaw and longitudinal	(-) 0.58 g (based on axle weight)	3g with weight transfer	Tyre contact patch
Pot hole	1.5 g	0	4 g	Tyre contact patch

Actual forces are calculated including dynamic effects (e.g. weight transfer for lateral acceleration) unless stated.

Sign Convention:

X =Positive rearward
Y =Positive to the right
Z =Positive upwards

Notes:

(1) Z direction loading includes 1g static load

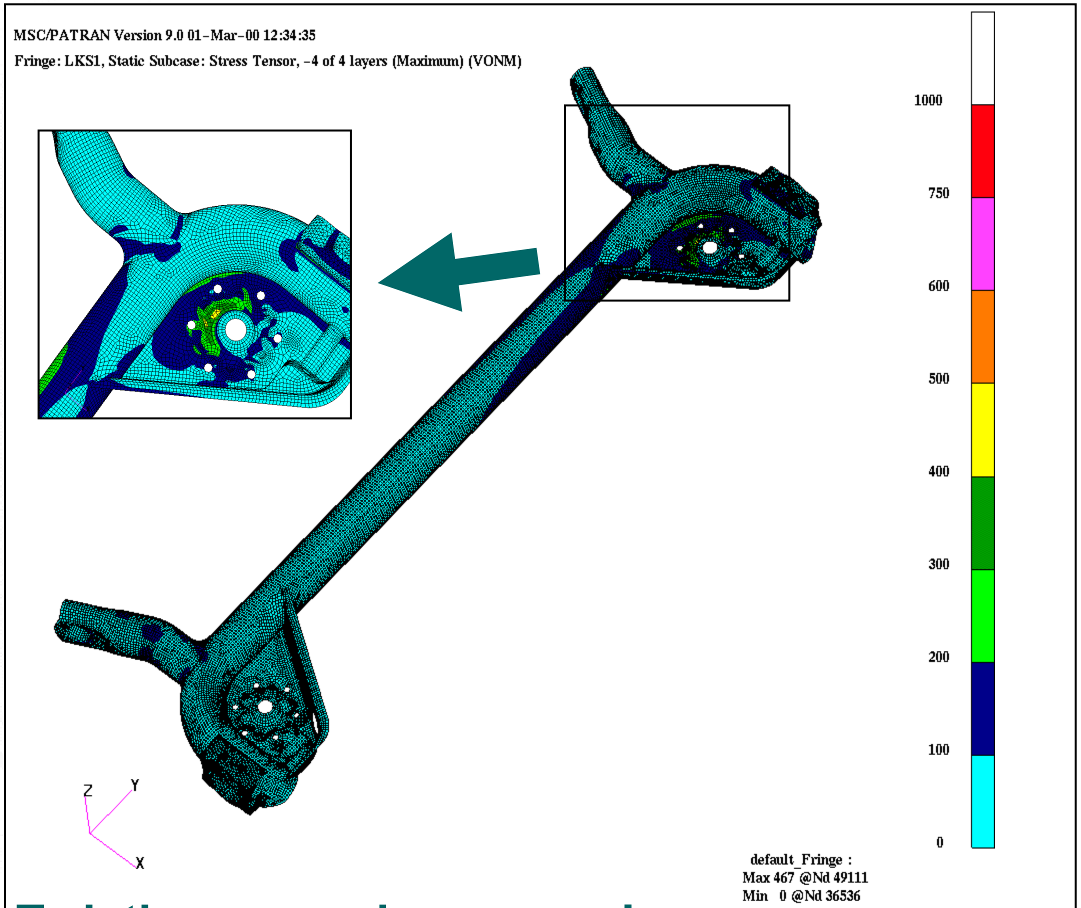
(2) Loads to be calculated assuming that the vehicle is in the Gross Mass condition.

Unit loads are applied to the FE models at the tyre contact patch and any other specific application areas. These are then combined to produce the standard proof load cases for stiffness and strength assessment. The proof load cases are obtained from Lotus' in house software and are as follows:

Reverse Curb Strike.
Lateral Curb Strike With Load Transfer.
Lateral Curb Strike Without Load Transfer.
Vertical Bump.
Forward Braking With ABS.
Combined Bump And Cornering.
Pothole Braking.

CAE STRUCTURAL APPROACH

Analysis



Twistbeam used as example

The two main types of analysis performed are linear static, and nonlinear static. For the nonlinear static analysis the nonlinear material model has to be specified, and the nonlinear load case must also be defined. (It is not possible to combine nonlinear static results.)

ULTRALIGHT STEEL AUTO SUSPENSION

CAE STRUCTURAL APPROACH

Results

Load Case	Max stress (Von Mises)	Location
Reverse Curb Strike (TCP)	468 MPa	Spring pan
Lateral Curb Strike 1 with load transfer	472 MPa	Spring pan
Lateral Curb Strike 2 with NO load transfer	416 MPa	Knuckle join
Vertical Bump (TCP)	592 MPa	Tube
Forward Braking with ABS (TCP)	355 MPa	Knuckle join
Combined Bump and Cornering (TCP)	445 MPa	Spring pan
Pothole Brake (TCP)	589 MPa	Tube

For the linear static analysis, after combining the unit load cases, the deformation of the FE model is checked to make sure the model is behaving correctly, and to obtain any stiffness values. The Von Mises stress value for each load case is then compared against the yield stress of the material. The element averaging definition domain should be compared between all entities and none. This gives an indication as to how good the mesh density and stress convergence is. If the stress value goes above the yield stress for very localised areas, this is acceptable. However, if there are considerable areas above the yield stress, then a the part design needs to be redefined. If this is not possible then nonlinear static analysis may be performed to further evaluate the behavior of the component under.

STEEL AUTO SUSPENSION

ULSAS CAE DYNAMICS APPROACH

Mechanical Dynamics Industries ADAMS software.

- System structural components are represented as Rigid Elements.
- Compliant joints are represented using ADAMS Bushing statements.
- Ball joints are represented using ADAMS Spherical Joint statements.
- Wheel bearing and strut bending compliances were represented using ADAMS Bushing statements.

The suspension geometries for the ULSAS systems were developed using Mechanical Dynamics Industries ADAMS software, version 9.1.

System structural components (links, arms, hub carriers, etc) were represented as rigid elements.

Compliant joints (bushes) were represented using ADAMS BUSHING statements.

Ball joints were represented using ADAMS SPHERICAL JOINT statements.

Wheel bearing, and where appropriate strut bending, compliances were represented using ADAMS BUSHING statements.

ULSAS CAE DYNAMICS APPROACH

Models were used to:

- Generate the overall characteristics of the suspensions with respect to vertical wheel displacement.
- Establish the contribution of both the structural and non-structural components of the system to the overall system compliance characteristics.
- The system geometry, stiffness of components and compliant joint stiffness were carefully tuned to obtain a solution which satisfied the programme kinematic and compliance targets.

Models were used to generate the compliance and kinematic characteristics of the suspensions with respect to vertical wheel displacement, and to establish the contribution of all of the components of the overall system performance. The system characteristics were established with respect to lateral and longitudinal forces applied at the tyre contact patch centre, and torque applied about a vertical axis through the tyre contact patch centre.

To obtain the maximum level of accuracy in the prediction of wheel displacements and rotations with vertical wheel displacement, the ULSAS Twistbeam has been simulated by Abaqus non-linear F.E. analysis. In general, ADAMS is used to analyse suspension systems, where at the concept stage the main suspension components are represented as rigid bodies, with bushes represented as linear characteristics. ADAMS generates a correct representation of the geometry changes during suspension system motion.

Structural compliance can be incorporated into an ADAMS model by using ADAMS beams. This is satisfactory at a basic concept level, but does not accurately model complex geometry. Alternatively, linear compliance of the parts can be included, if required, by using ADAMS F.E. This allows the rigid parts in ADAMS to be replaced by F.E models. These F.E. models are generated using an external code (Nastran etc.).

ULSAS CAE DYNAMICS APPROACH

Models were used to:

- Generate the overall characteristics of the suspensions with respect to vertical wheel displacement.
- Establish the contribution of both the structural and non-structural components of the system to the overall system compliance characteristics.
- The system geometry, stiffness of components and compliant joint stiffness were carefully tuned to obtain a solution which satisfied the programme kinematic and compliance targets.

Systems that require the structural compliance to be included can be modelled directly using F.E. analysis. Twistbeams fall into this category. If non-linear geometry effects are to be calculated (as they should be) non-linear F.E. analysis should be used (ABAQUS etc.) This type of analysis can also be used to represent a pseudo-kinematic system, with bushes and joints – and will give identical kinematic and compliant results to ADAMS F.E., where structural deflection of parts is small.

For twistbeams, structural compliance generates non-linear geometry changes. ADAMS F.E. would not produce the correct answer over large deflections (such as roll simulation). ADAMS can only support linear structural compliance, valid only for small deflections. For large structural deflections, as in a twistbeam kinematic analysis, non-linear structural compliance must be represented for accurate results, was achieved using a non-linear F.E. tool. As F.E. models already existed (for stress results etc.), Abaqus was used to analyse the Kinematic and Compliant characteristics. An ADAMS model could have be created in addition, but would have been of no benefit. The system geometry, component stiffness and compliant joint stiffness were all varied to obtain a solution which satisfied the kinematic and compliance targets generated by the target setting process.