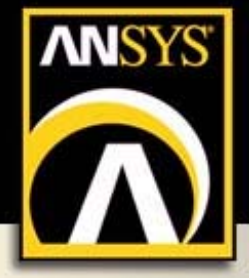
2006 International ANSYS Conference
May 2-4, 2006

Development of a Finite Element Model to Analyse the Proximal Ring of an Endovascular Device

Emma McCummiskey
Department of Mechanical Engineering
University of Strathclyde
Glasgow, UK.

Overview



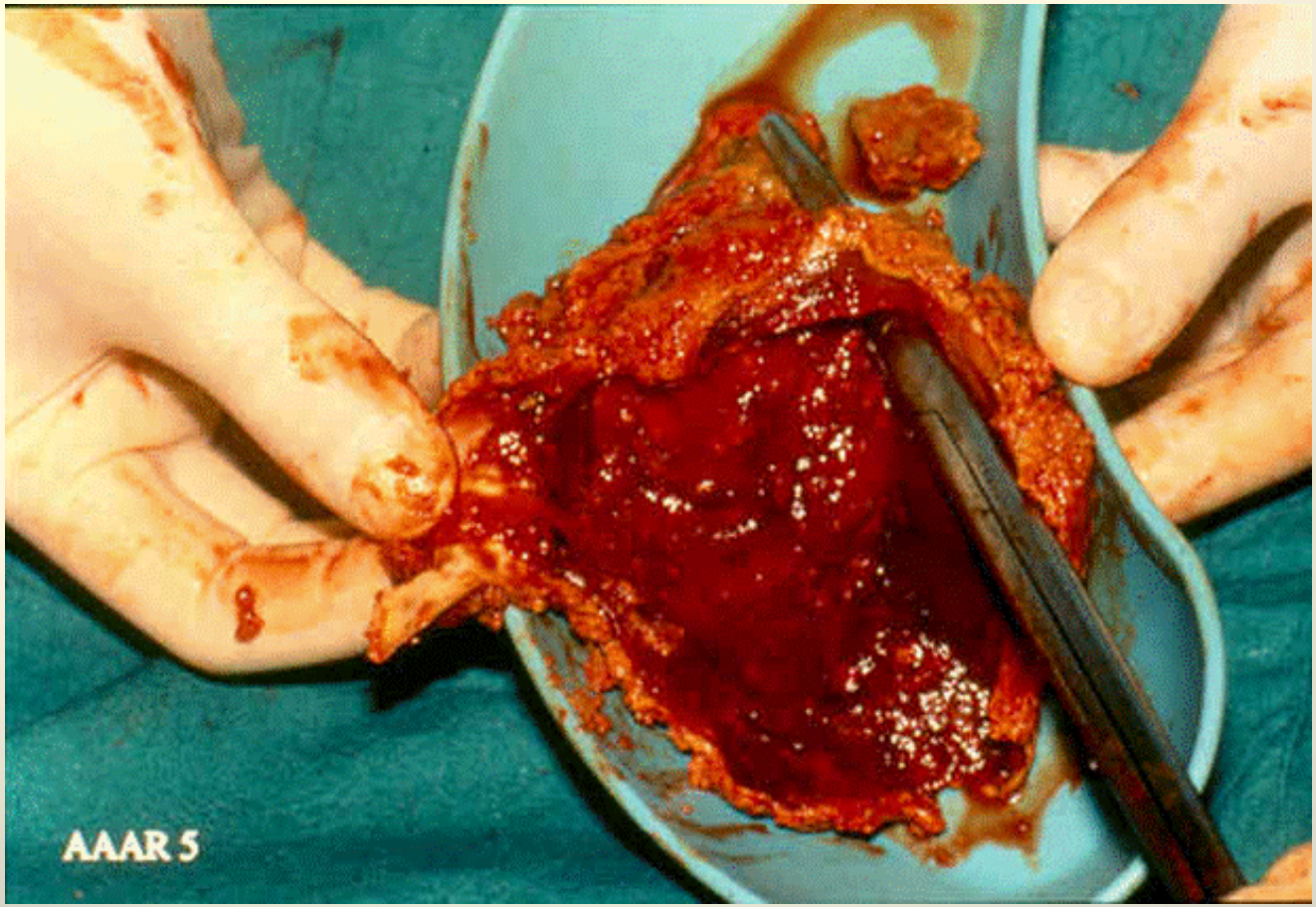
- Medical background
- Modelling the device with finite element analysis
- Complexities of modelling
- Results
- Future work
- Questions

What are AAA's?

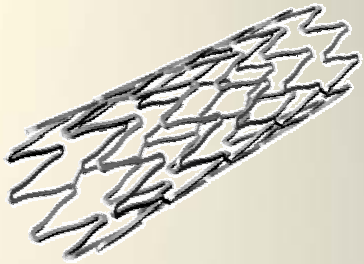
- Local swelling in abdominal aorta
- Known as abdominal aortic aneurysms
- A major cause of death in western world
- If ruptured only 25% of patients will survive
- Most patients are high risk for remedial surgery

Aneurysms

Ruptured Aneurysm



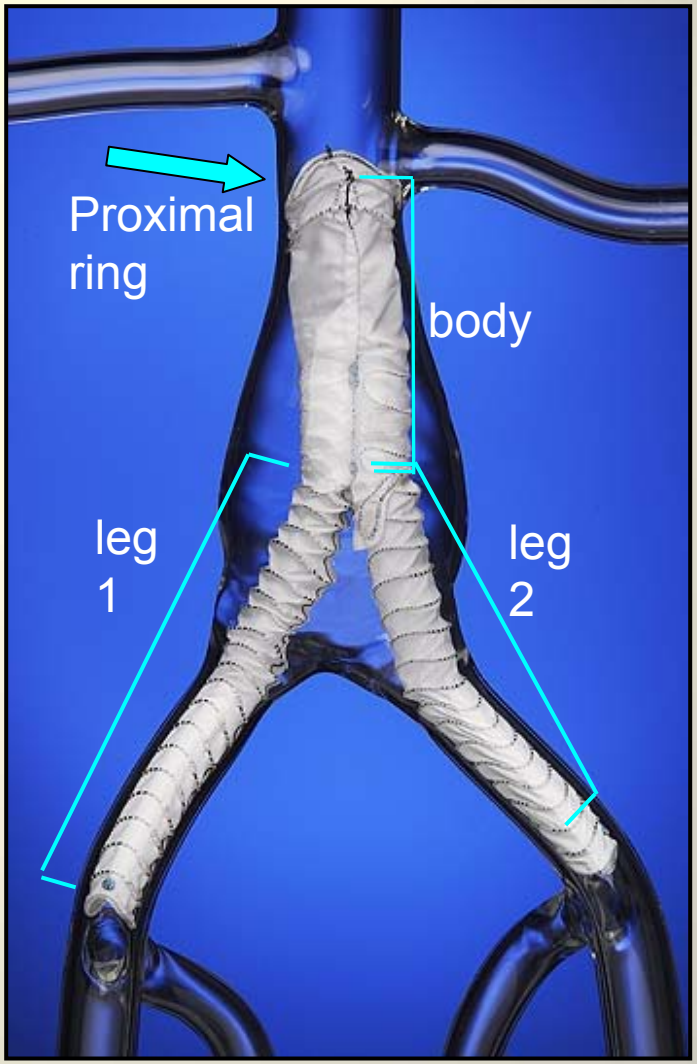
Modelling



- Most work to date on cardiovascular stents
  - Different structure and purpose
- Difficult to define strain state which is heavily relied upon for fatigue
- Most important region is the **proximal ring** as this is relied upon for sealing
  - In this work, only the proximal ring will be modelled.

Endovascular Repair

- Self expanding graft of 3 discrete modules
- Two small incisions in the femoral arteries
- Catheter inserted and deployed into position
- More complicated engineering design

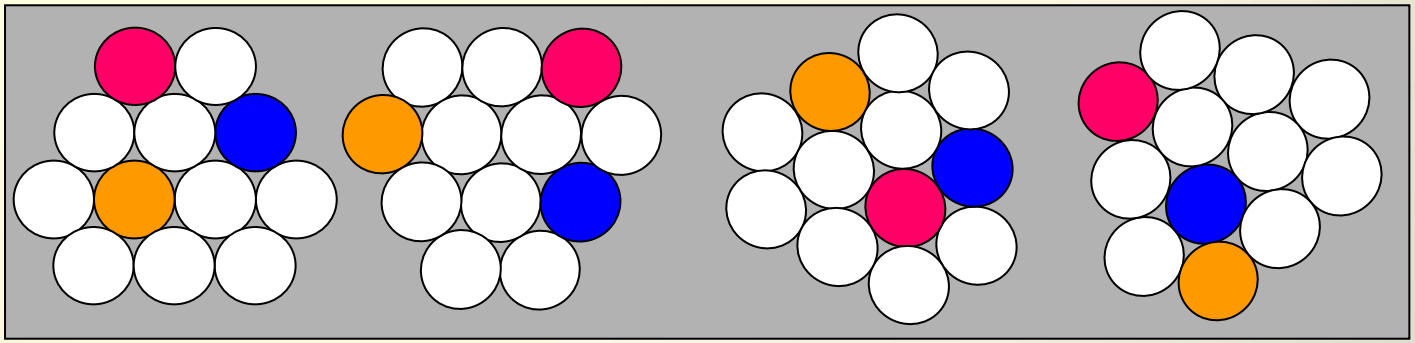


Modelling issues

- Geometry and bundle
- Material model
- Contact issues
- Pre-strain in the model

Geometry and wire bundle

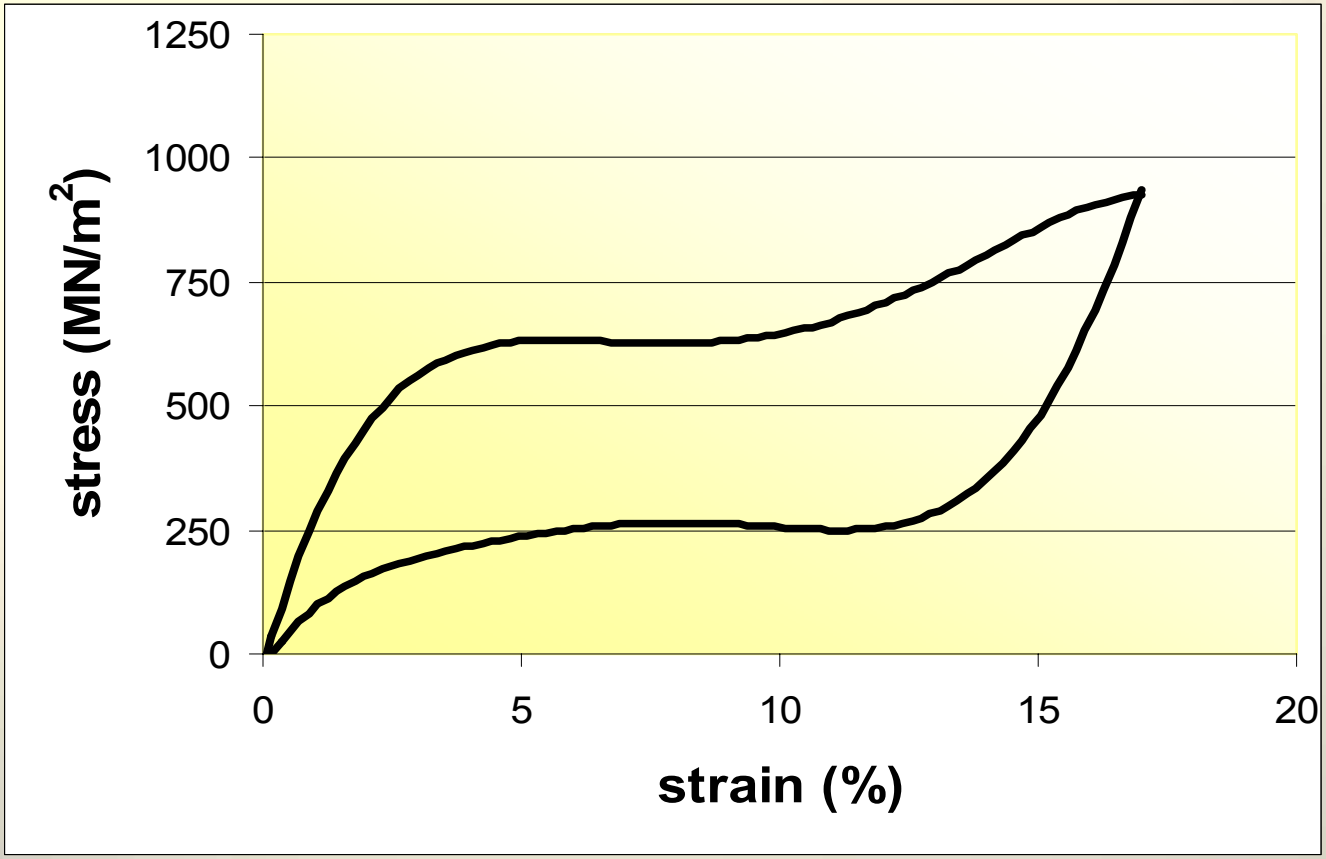
- Multiple turns of wire wound into a ring and sutured into fabric in a slight saddle
- Difficult to ascertain the configuration of wires around the ring



- In this work a single representative wire will be modelled
- Model is symmetrical about two planes thus can be modelled as a ¼ model

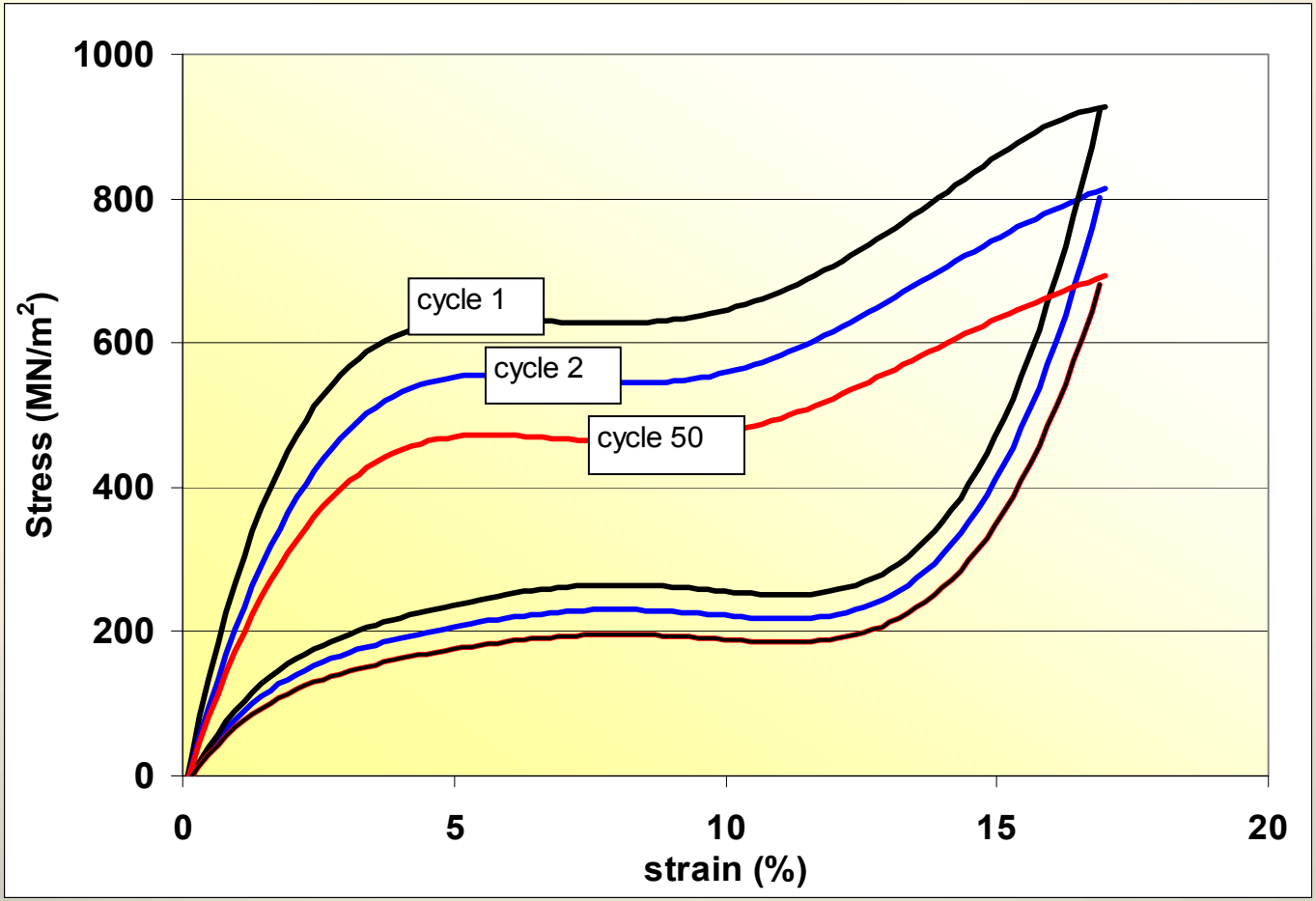
Superelastic Nitinol

- Can undergo very high strains (8-10%) without permanent deformation



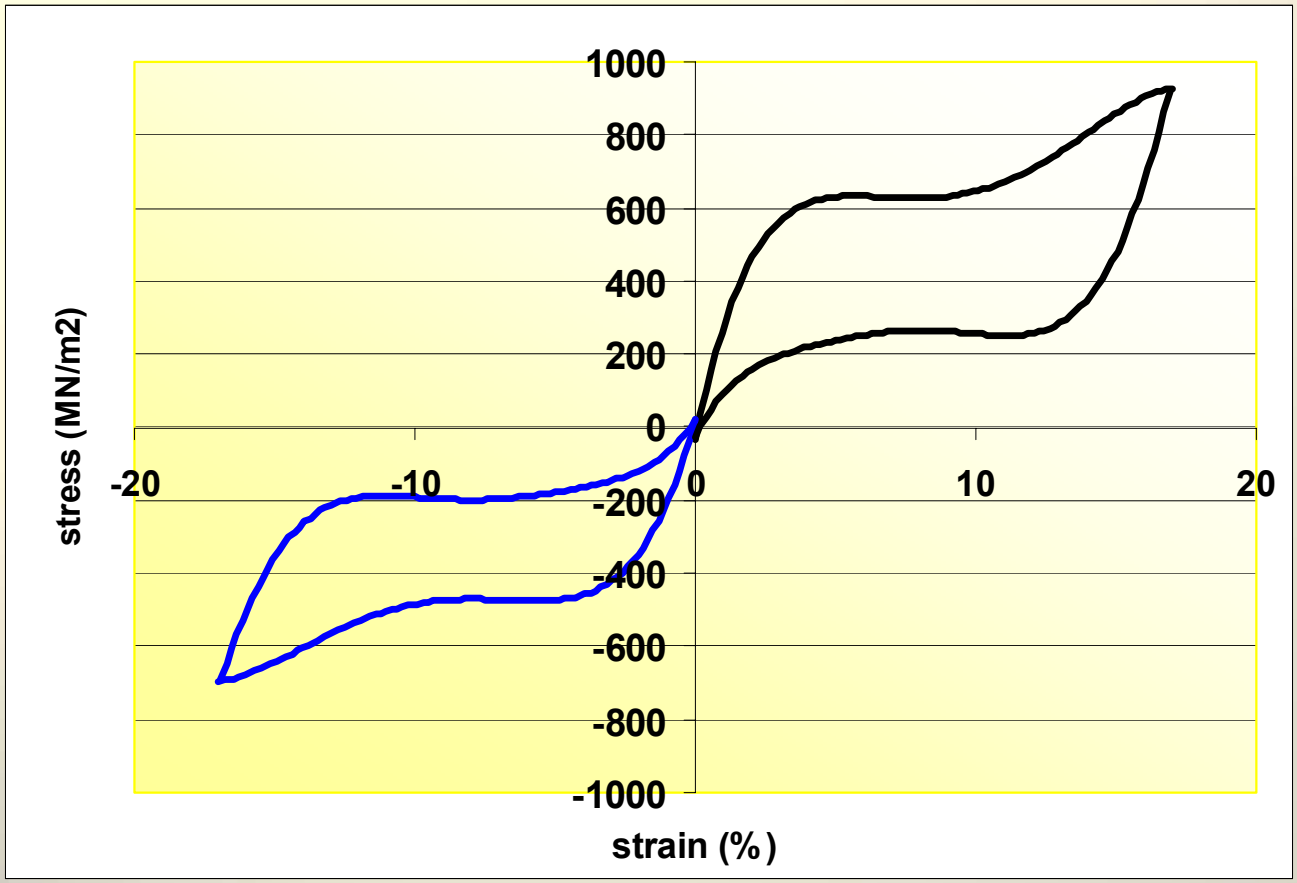
Superelastic Nitinol

- Is subjected to cyclic softening

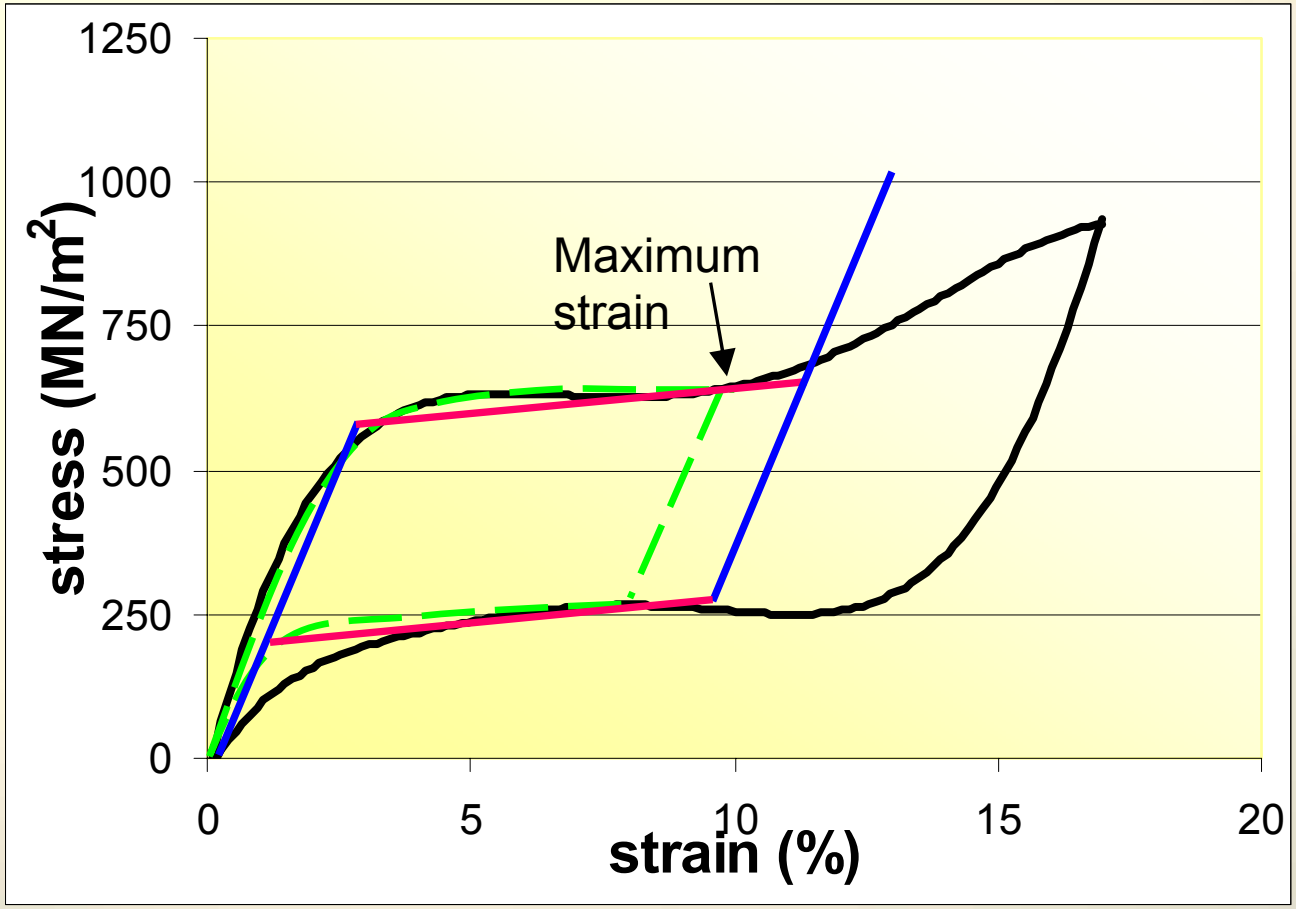


Superelastic Nitinol

- Exhibits different behaviour in tension and compression



Nitinol Material Model



- Ansys approximates the curve as shown

Contact Modelling

- Used to determine the stresses experienced by the artery
- Large contact stresses may cause penetration or cutting
- Low contact stresses may cause inadequate sealing
- Both surface to surface and node to surface contact used in simulation

Pre-strain in the model

- Use the proximal ring is pre-strained during the manufacturing process
- Structural pre-strain analysis modelling straight wire, roller and former
- Roller rotated around the former to bend the wire thus creating accurate strain distribution
- Sequential solution for deployment in tube with pre-strain

Applying the Pre-strain

Create ¼ model geometry in Ansys

Use Ansys approximation of Nitinol material

Apply appropriate constraints and contact pairings to the geometry and bend wire through 90 degrees

Store displacements and apply to each node.

Remove the Roller and former

Deployment Analysis

Apply symmetry to wire and tube

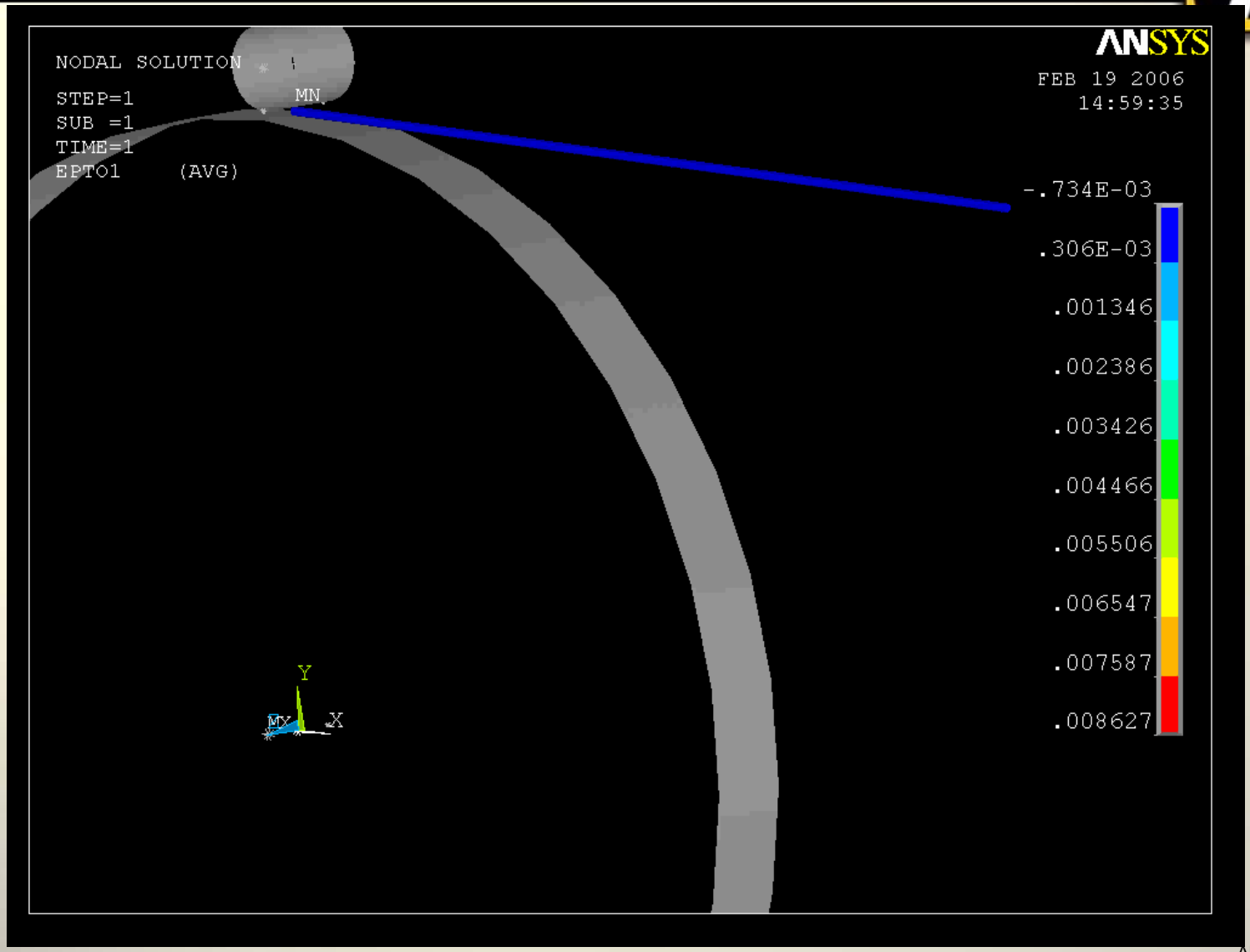
Deform the wire longitudinally into1/4 saddle

Insert wire into tube

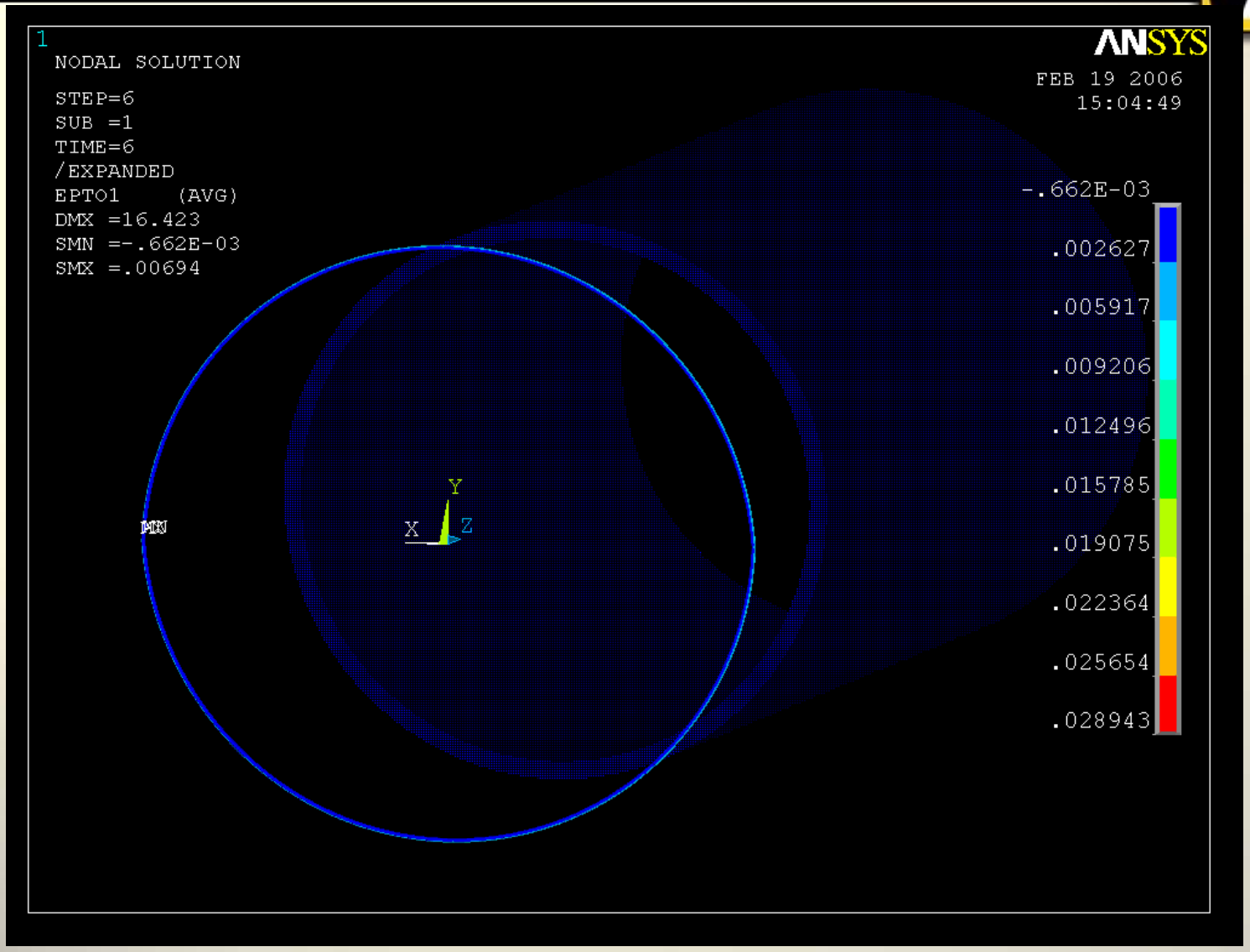
Remove displacement from wire and allow to settle into tube.

Analyse results to determine equilibrium position

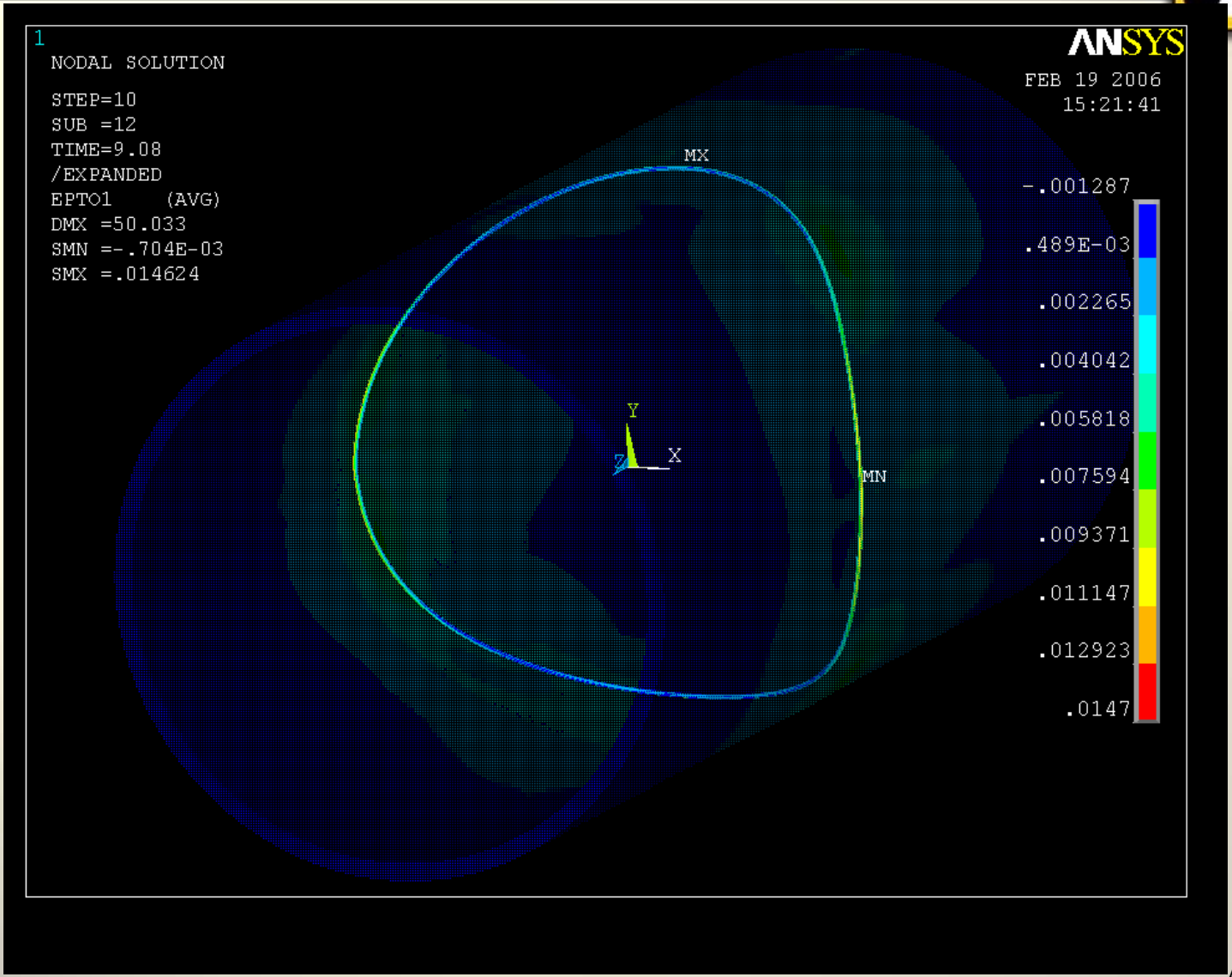
Pre Strain Analysis



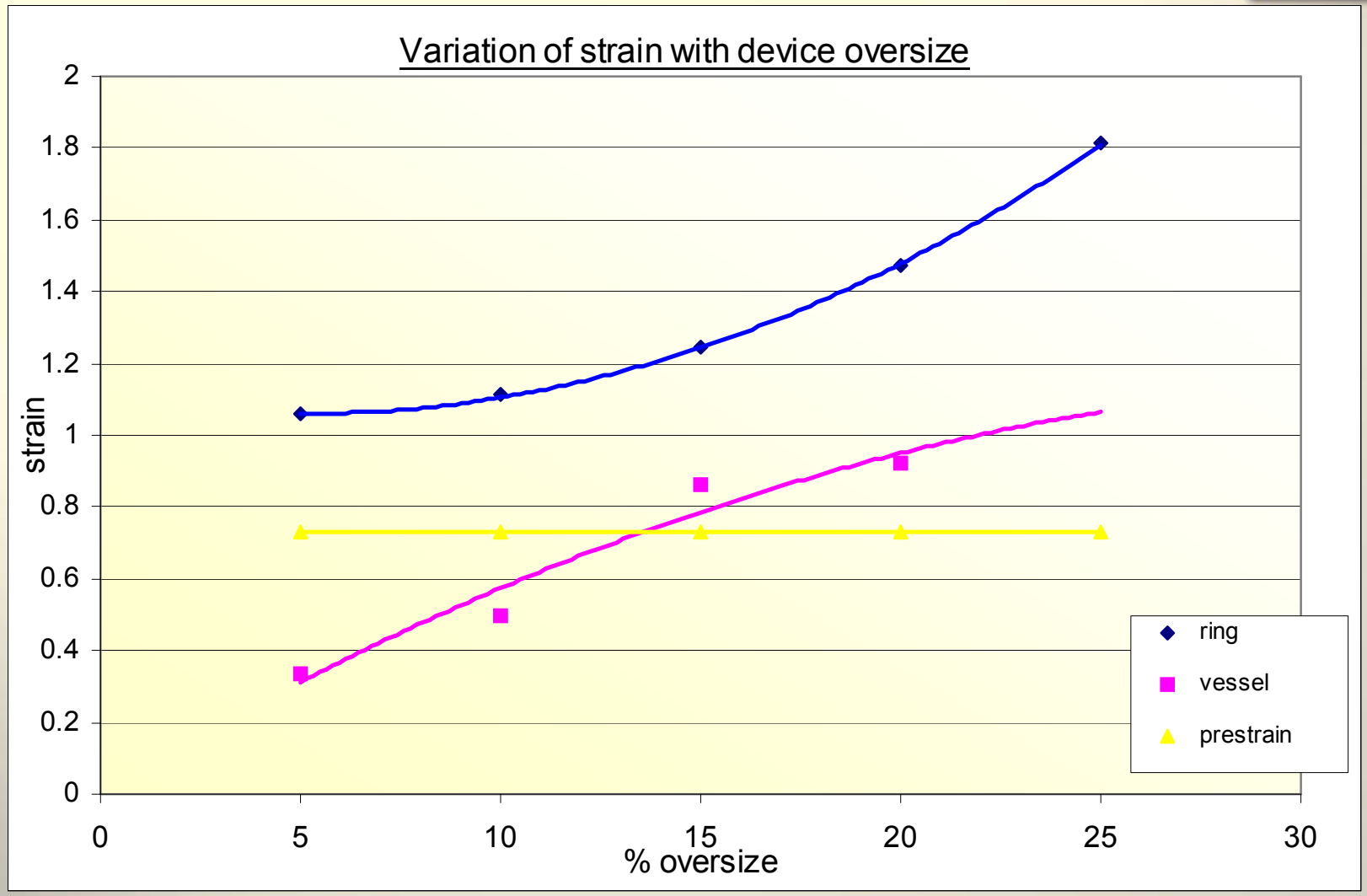
Compaction Analysis



Deployment Analysis



Variation of Oversize on a B30



Conclusions

- The Nitinol material model provided by ANSYS is adequate for our purposes
- The pre-strain, compaction and deployment can be modelled in ANSYS
- The maximum strain undergone by the ring is ~1.8%

Future Work

Future work will include

- Validation of numerical findings
- Investigation of bundle model
- Model of the artery

Acknowledgements

The authors would like to acknowledge

- the use of ANSYS and under a university research license
- Ansys UK support
- Funding from the Medical Devices Faraday Partnership
- Funding from Terumo Vascutek